ZondTEM3D

Software for three-dimensional interpretation of transient electromagnetic (TEM) data. User manual

2026

Contents

About the program 6
Purpose 6
What is needed for a calculation 6
Units and notation 7
Program modules 7
System requirements and installation 9
Supported configurations 10
Transmitters 10
Receivers 10
Configuration layouts 10
Measured quantities and current waveform 11
Main window 13
Layout of elements 13
Toolbar 14
Status bar 15
Mouse and keys of the main window 16
Opening data and projects 17
Supported formats 17
Sequence of dialogs on opening 18
Reader module dialogs 20
Stations locations window 22
Data filter window 23
Line settings window 25
Station parameters window 27
Workflow: loading field data 30
Viewing and processing data 31
Data display modes 31
Sounding curves window 32
Data smoothing 33
Inversion data norm (Lin-log threshold) 34
Removing data with large misfit 35
Rx-Tx alignment along lines 36
Other data operations 36
Workflow: data quality control 36
Main menu 38
File menu 38

Options menu 38
Buffer menu 42
Help menu 42
Inversion button pop-up menu 43
Mesh constructor 44
Purpose 44
Automatic selection of the vertical mesh 44
XY view tab 45
XYZ view tab 47
Trapezoidal mesh and Simple grid maker windows 48
Messages when creating the mesh 48
Workflow: building the mesh 48
Model and its editing 49
Model cells, parameters and constraints 49
Start model and constraints dialog 49
Set apriori boundaries window 52
Model editing 55
Forward problem and inversion 61
Forward problem 61
Inversion types 61
Program setup window 63
RMS vs iteration window 70
Misfit evaluation of inversion results 70
A priori information 71
Graphic backgrounds 71
Model build-in 72
Borehole data 73
Interpretation results 79
Model slice 79
Volume window 79
3D settings window 80
Opacity depending on the parameter 85
Model smoothing and rasterization 86
Model cutting 87
Several models in a project 88
Saving results 89
Building slices in Surfer 89
Workflow 90
Mouse and keys of this chapter 90

Graphic settings 91
Contour map and pseudosection setup dialog 92
Palette setup dialog 96
Curve set editor 97
Plot editor 99
Axis editor 100
Print preview dialog 101
Auxiliary windows 103
Project information window 103
Help info window 104
Send message window 104
Keys and mouse in graphic settings windows 105
Typical work scenarios 106
Full cycle: from field data to a 3D model 106
Data quality control 107
Modelling and resolution check (synthetic test) 108
Single-line survey (2D mode) 108
Areal survey with several lines 108
Using a priori information 109
Appendix A. File formats 110
TEM data 110
Mesh and topography 114
Models: mod3d, mod2d, XYZV, XYZ slices 114
Boreholes 119
Palettes and .clr levels 120
Export: Paraview, Oasis, Surfer/Voxler, XYZ, KML 122
Other service files 124
Appendix B. Keyboard shortcuts and mouse actions 125
Main window and graphic zones 125
Model and slice editor 126
Sounding curves window 127
Mesh constructor and Trapezoidal mesh 127
Lines, stations and data filter 128
Pseudosections and maps 129
3D volume window 129
Graphics and print windows 130
Other windows 130
Appendix C. Program files and modules 132
Files in the program folder 132

Temporary exchange files 132
Program modules and windows 133

About the program

Purpose

ZondTEM3D is a program for three-dimensional modeling and inversion of time-domain electromagnetic data: near-zone transient soundings and the transient electromagnetic method (TEM/TDEM). The program computes transient responses over a three-dimensional earth with an arbitrary distribution of electrical resistivity (including topography) and solves the inverse problem, i.e. recovers a three-dimensional resistivity model from the observed data.

The program is intended for areal surveys and networks of profiles: surveys with coaxial and central loops, offset configurations (one transmitter loop and many receivers), coincident loops, as well as data from airborne and towed systems (for example, TTEM and STEM). Import of the most common field instrument formats is supported; the program itself converts all data to a uniform form (signal normalized to μV/A, time in seconds) and to a single coordinate system.

Main features:

- import of TEM-FAST, ERA, Podbor, ZondTEM1D (tdf, usf, avg), Interpex, Geonics, Amira, BEM, STEM and TTEM data; merging several files into one project;
- editing the configuration of each station: shape and position of the transmitter and receiver loops, receiver type, current waveform (step, trapezoid with ramps, arbitrary shape);
- quality control: pseudosections and maps of apparent resistivity and signal, sounding curves, smoothing, data filter, removal of points by misfit;
- construction of a three-dimensional mesh from station positions, with topography and cell grouping;
- fast quasi-three-dimensional inversion (**Pseudo 3D**): coupled one-dimensional inversions beneath the stations with constraints between neighbours and transfer of the result to the three-dimensional mesh;
- full three-dimensional inversion (**Full 3D**) based on the finite element method (Nédélec edge elements) and the rational Krylov method;
- model editor, layered starting model, parameter limits, background images, boreholes, model cutting;
- three-dimensional visualization of the result, export to Surfer, Voxler, ParaView, Oasis montaj, exchange formats MOD3D, MOD2D, XYZV.

What is needed for a calculation

To solve the forward and inverse problems the program needs:

1. **Survey layout**: station coordinates, geometry of the transmitter (Tx) and receiver (Rx) loops, loop elevation above the surface, receiver type, current waveform and the list of times (delays).
2. **Earth model**: a three-dimensional mesh of cells with the resistivity of each cell, and the surface topography.
3. **Observed data** (for inversion): EMF values normalized to current at each time of each receiver and (optionally) measurement weights.

When reading a field file the program goes through a chain of dialogs in which missing information (units, coordinates, profiles) is specified, and then builds the mesh. The procedure is described in the section “Opening data”.

Units and notation

Quantity	Units in the program	Note
Time (delay)	s	In the pulse shape dialogs durations are specified in microseconds (μs)
dB/dt signal	μV/A	EMF normalized to the transmitter current; optionally also to the loop areas (see Data units)
Field B	value of B normalized to the transmitter current (B/I)	Used if the dF/dt flag of the receiver is cleared (field B instead of dB/dt); normalization to current is the same as for the dB/dt signal
Apparent resistivity ρa	Ohm·m	By default, the late-time asymptote; on request, the “true” ρa of a homogeneous half-space
Coordinates, elevations	m	Z is the absolute elevation, H is the height above topography
Model resistivity	Ohm·m	Allowed range 0.0001–1 000 000 Ohm·m

Late-time apparent resistivity. For a dB/dt receiver the program uses the formula

$$\rho_\tau = \frac{\mu_0}{\pi t}\left(\frac{q\,\mu_0}{20\,t\,|u|}\right)^{2/3},$$

where t is time, s; u is the signal normalized to current (V/A); $q = S_{Tx}\,S_{Rx}$ is the product of the effective areas of the transmitter and receiver loops, m²; $\mu_0 = 4\pi \cdot 10^{-7}$ H/m. The formula coincides with the classical $\rho_\tau = \frac{\mu_0}{4\pi t}\left(\frac{2\mu_0 q}{5tu}\right)^{2/3}$. For the magnetic field B the analogue $\rho_\tau = \frac{\mu_0}{\pi t}\left(\frac{q\,\mu_0}{30\,|B|}\right)^{2/3}$ is used. If a long turn-off ramp is specified, a version of the formula with a ramp correction is applied.

True apparent resistivity (item **Options / Data / True apparent resistivity**) is the resistivity of a homogeneous half-space for which the signal computed for the full configuration (loop shape, elevation, ramps) equals the observed one at the given time. It reflects the section more accurately at early times, but takes longer to compute and is used only for display: smoothing, filtering and inversion norms work with the late-time ρa or with the signal.

Program modules

The program consists of the main window and a set of windows (modules), each responsible for its own stage of work. The modules are listed below with a brief purpose; a detailed description is given in the corresponding sections.

Module (window title)	Purpose	How to open

Module (window title)	Purpose	How to open
ZondTEM3D (main window)	Data map, 3D view, pseudosections, model editor, running calculations	—
Amira / Geonics survey / Data units / UTM zone	Specifying units, normalization and parameters when reading a file of a particular instrument	Automatically on opening
Curve selector	Selection and averaging of repeated curves of one sounding (Deep processing)	Automatically on opening
Stations locations	Table of station coordinates, loading coordinates from a file, conversion of latitude/longitude to UTM	Automatically on opening
Data filter	Removal of data outside ranges, by polygon, station thinning	On opening (if **File / Show data filter** is enabled)
Line settings	Creating and editing survey profiles (lines)	On opening, **Options / Set survey lines**
Station parameters	Station parameters: coordinates, transmitter, receiver, current waveform	Button 
Pulse shape	Tabulated current waveform	Button in **Station parameters / Time mode**
Mesh constructor	Construction of the three-dimensional mesh and topography	On opening, **Options / Mesh constructor**
Sounding curves	Sounding curves of the selected receiver, editing points	Button , **Options / Data / Sounding curves**
Smoothing settings	Smoothing of observed data in time and between stations	**Options / Data / Smooth data**
Program setup	Inversion parameters	Button , **Options / Program setup**
Start model and constraints	Layered starting model and limits	**Options / Model / Set 1D model**
Set apriori boundaries	A priori boundaries	**Options / Inversion / Set apriori boundaries**
Extra layered term	Layered term settings	**Options / Inversion / Optimization / Layered term**
RMS vs iteration	Inversion progress	Automatically (**Options / Inversion / Show progress**)
Data misfits	Misfit distribution, removal of records with large misfit	**Options / Extra / Remove data with big misfit**
3D settings	3D view settings	Button , right click on the 3D view
Cell setup / Model setup	Exact cell values, model editor view	Model editor menu
Borehole editor	Boreholes, lithology, logs	**Options / Boreholes**
Buffer	Comparison of up to four models	Menu **Buffer**

Module (window title)	Purpose	How to open
Graphics setup windows	Contours, palettes, axes, printing, video	Right click on plots

The computational part is divided into two parts. The fast quasi-three-dimensional inversion (Pseudo 3D) is performed inside the program. The full three-dimensional forward problem and Full 3D inversion are performed by a separate 64-bit computational module **zondtem3d.dll**. If the module is not found, the message “zondtem3d.dll not found” is displayed at startup. If calculations are constantly interrupted, check whether an antivirus blocks the computational module, and run **Help / Restart library**.

System requirements and installation

The program runs on 64-bit Windows 8–11. For full three-dimensional inversion a multi-core processor and at least 64 GB of RAM are recommended: the amount of memory is determined by the number of mesh cells and the number of inversion parameters (the current estimate is shown in the status bar). For fast work with large meshes use cell grouping (**Cell grouping**) and model cutting.

To install, copy the program folder to a disk where writing is allowed (service files for exchange with the computational module are written to the program folder). The license is bound to a hardware key or a registration file (**File / Reg file**).

Supported configurations

ZondTEM3D works with inductive TEM configurations with ungrounded sources. Only the vertical (Z) component of the magnetic field or its time derivative is used. The configuration is specified separately for each station (window **Station parameters**), so stations with different transmitter and receiver geometry can be combined in one project.

Transmitters

Transmitter type	Code in TDF	Description
Loop: loop	3	Ungrounded loop of arbitrary shape: square, rectangle or polygon given by its vertices (the contour is closed automatically). The loop elevation above the surface is set by the **Elevation** parameter: 0 for a ground loop, greater than 0 for a loop of a towed or airborne system
VMD: vertical magnetic dipole	0	Small loop whose size is much smaller than the offset and the depth of investigation; the centre and the elevation above the surface are specified

Grounded lines (HEL), horizontal electric (HED) and magnetic (HMD) dipoles are not supported: stations with such transmitters are excluded from calculation and inversion.

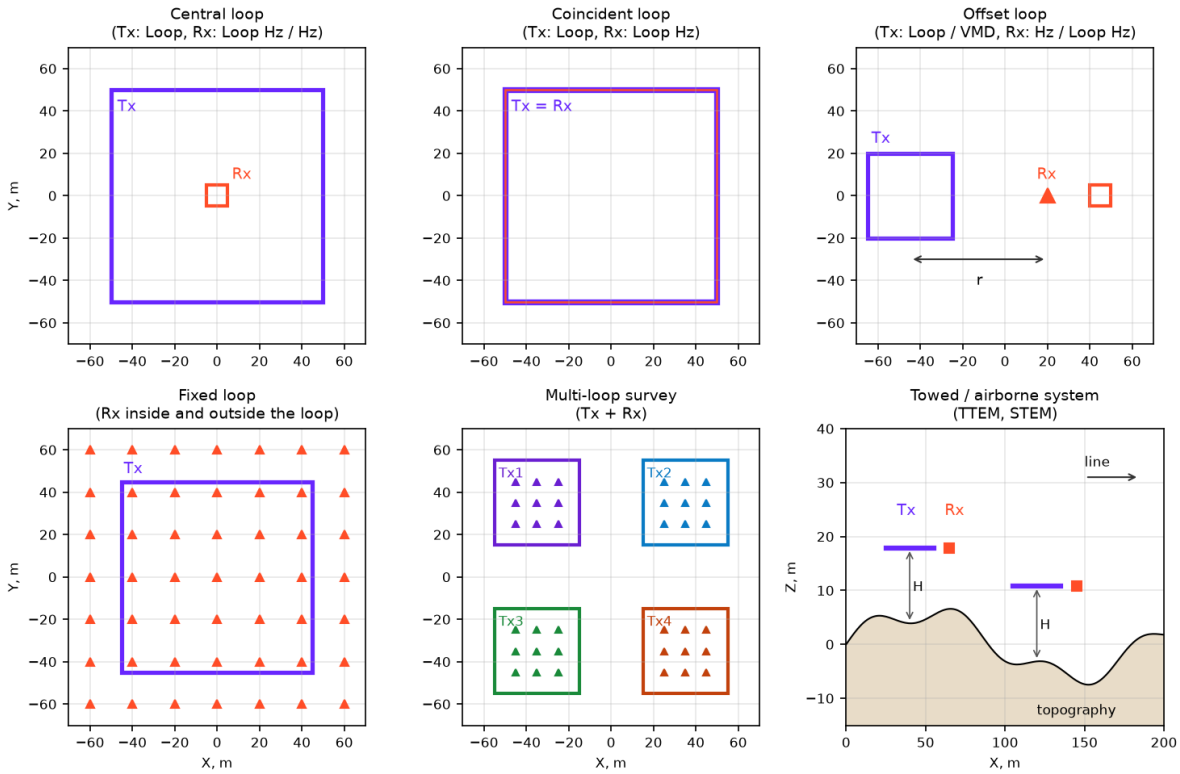
Receivers

Receiver type	Code in TDF	Description
Loop Hz: receiver loop	2	Loop of arbitrary shape; the response is computed along its contour taking into account the area and the order of vertices
Hz: vertical magnetic dipole	3	Point induction sensor of the vertical component; the effective area is taken into account in data normalization

Horizontal components (Hx, Hy) and electric receivers (Exy, MN) are skipped on reading, and components are not combined into a total field; stations with no Z receivers left are not loaded. A station may have several receivers, i.e. a group of receivers associated with one transmitter loop. The elevation of each receiver above the surface is specified separately.

Configuration layouts

All common layouts of ground, towed and airborne surveys are assembled from the transmitters and receivers listed above.



Supported configuration layouts

Layout	Transmitter	Receiver	Features
Central loop (in-loop)	Loop	Loop Hz or Hz at the loop centre	The main layout of areal soundings; the transmitter and receiver move together
Coincident loop	Loop	Loop Hz coinciding with the transmitter loop	The transmitter loop serves as the receiver loop at the same time
Offset loop	Loop or VMD	Loop Hz or Hz outside the loop	The receiver is offset from the transmitter by a given separation and moves together with it
Fixed loop	Loop	Group of Loop Hz or Hz inside and outside the loop	One large loop for many receiver points; receivers of one loop are highlighted on the map by Ctrl + click
Multi-loop survey	Several Loops	Own group of receivers for each loop	Loops with identical geometry and elevation are merged: the field of each transmitter loop is computed once for all its receivers
Dipole–dipole (small loops)	VMD	Hz	Small dipole loops at a given separation
Towed and airborne systems (TTEM, STEM)	Loop at an elevation	Loop Hz or Hz at an elevation, with an offset	Transmitter and receiver elevations are specified for each station; surface topography is taken into account

Measured quantities and current waveform

Parameter	Options
Measured quantity	dB/dt: EMF normalized to the transmitter current, μV/A (receiver **dF/dt** flag on); B: magnetic field normalized to current (flag off)

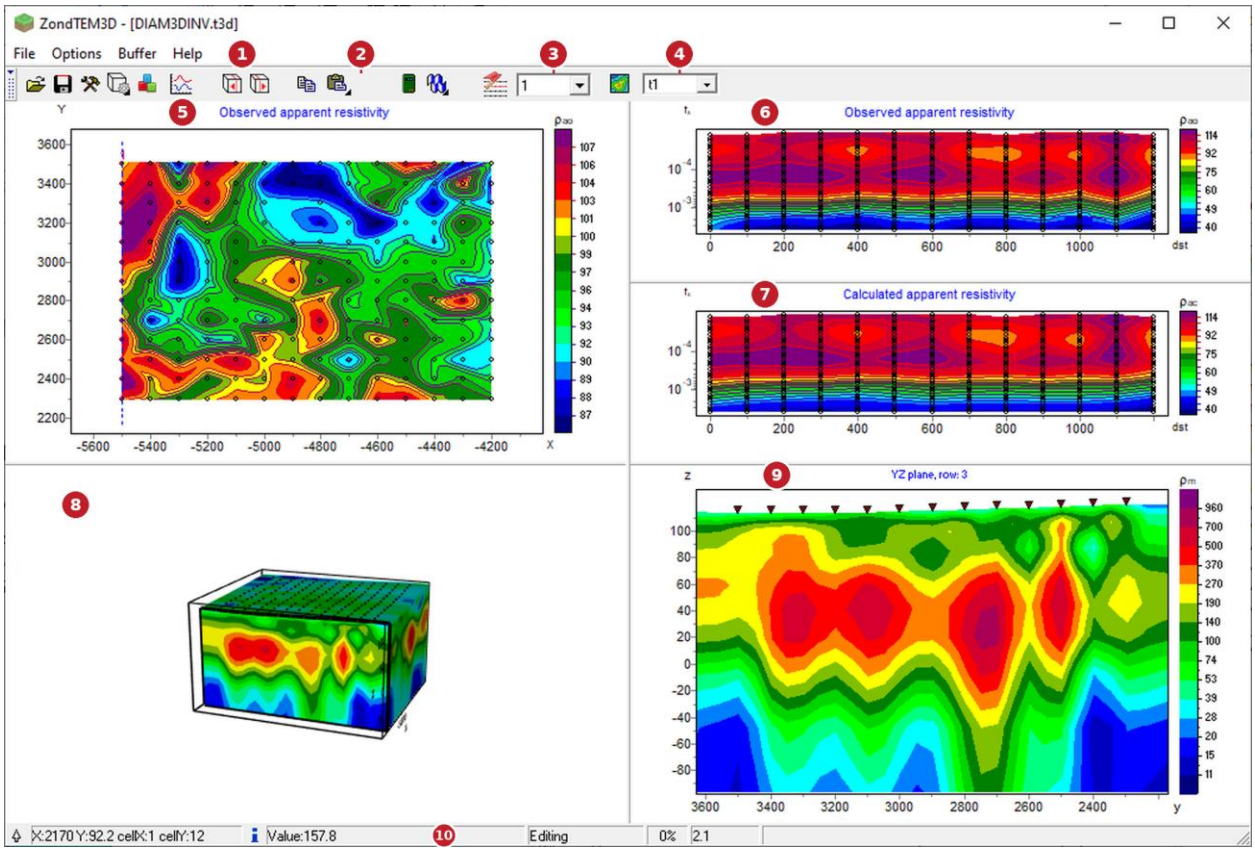
Parameter	Options
Current waveform	Step (instantaneous turn-off), finite pulse with turn-on and turn-off ramps (trapezoid), user-defined pulse shape
Current modes	Several modes (sweeps) per station, each with its own pulse shape and durations; each data sample belongs to its own mode
Survey type	Regular: a system of parallel profiles; General: an irregular areal survey; One line: a single profile

Field instrument formats from which data of these configurations are read are listed in the section "Supported formats". The configuration of each station is viewed and edited in the **Station parameters** window.

Main window

Layout of elements

After loading the data and building the mesh, the main window is divided into five graphic zones: top left, the data map; on the right, the observed and calculated pseudosections of the active profile; bottom left, the 3D view of the model; bottom right, the model slice (model editor). The numbers in the figure correspond to the rows of the table.

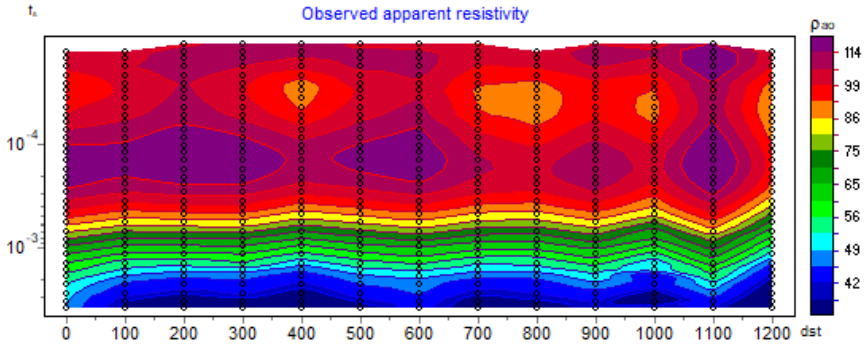


ZondTEM3D main window (numbers are window elements, see the table)

Element	What it shows	Main actions
1 — Main menu	Commands File, Options, Buffer, Help	See the chapter “Main menu”
2 — Toolbar	Buttons of the main commands: files, settings, stations, curves, slices, buffer, calculation and inversion	See the table of buttons below
3 — Line list	Active profile along which pseudosections 6 and 7 are built	Profile selection; profiles are defined in the Line settings window
4 — Time list	Time shown on data map 5	Time selection; right click sets the number of neighbouring times
5 — Data map (top left)	Stations and data values at the selected time: contour map (Contour map) or coloured points (Point map); profiles, the active profile and the selected receiver	Clicking a point selects the receiver

Element	What it shows	Main actions
6 — Observed pseudosection (top right)	Observed data along the active profile: horizontally, distance along the profile (dst); vertically, time (logarithmic scale). In Graphics plot mode, plots versus time	Clicking a point selects the receiver and opens its curves
7 — Calculated pseudosection (centre right)	Calculated data or misfit (Data misfit) in the same form	Same
8 — 3D view (bottom left)	3D model, stations, boreholes, background images, plane of the current slice	Left button rotates; wheel zooms; middle button pans; right button opens the 3D settings window
9 — Model editor (bottom right)	Slice of the three-dimensional model (XZ, YZ or XY); triangles are stations	Drawing, selection, wheel moves to the neighbouring slice; see the section “Model editor”
10 — Status bar	Cursor coordinates and cell number, parameter value, mode, calculation progress, RMS	See the table “Status bar”

The borders between zones are moved with the mouse. Sizes and proportions are saved when the program is closed.



Apparent resistivity pseudosection along a profile: circles are data points, the time axis is logarithmic

How to read a TEM pseudosection. Early times (top) characterize the upper part of the section, late times (bottom) the deep part: the penetration depth of the field grows approximately as $\sqrt{t\rho}$. An isometric anomaly of increased ρa at intermediate times in the figure corresponds to a conductive or resistive body at depth, and a vertical band of reduced values corresponds to a near-surface inhomogeneity or noise at one or two stations. Vertical “bands” that do not change with time often indicate errors in the loop area or in the signal normalization of a station.

Toolbar

Button	Purpose
	**Open file**: open a project (`*.t3d`) or a field data file. Same as **File / Open file**
	**Save file**: save the project or export data and model (the type is selected in the save dialog). Same as **File / Save file**
	**Program setup**: inversion parameters (algorithm, smoothing, resistivity limits, cell grouping)

Button	Purpose
	Menu of 3D view and model settings (the same menu is opened by right click in the 3D view via the **3D settings** window)
	**Station parameters**: parameters of the current station: coordinates, transmitter and receiver loops, current waveform
	Show or hide the sounding curves window (**Sounding curves**) of the selected receiver
	Previous / next model slice along the current direction (XZ, YZ or XY)
	Copy the model of the current slice to the internal buffer
	Paste the copied slice into the current one. Right click opens the paste menu: **Current** (current slice only), **Current&+** (current and all following), **Current&-** (current and all preceding)
	**Calculate forward problem**: compute data for the current three-dimensional model (full 3D forward problem by the finite element method in the computational module)
	**Inversion**: start the inversion; during calculation the button changes its look and pressing it again stops the calculation. Right click selects the inversion type: **Pseudo 3D inversion** (default) or **Full 3D inversion**
Line list	Active profile (line). Profiles are created in the **Line settings** window; if the file has none, they are built automatically
Time list	Time displayed on the data map (t1, t2, … is the time number; the exact time value is shown in the hint). Right click sets the number of neighbouring times (0–5) shown on the map together with the selected one

Status bar

Section (left to right)	Contents
Coordinates	Cursor coordinates over the model editor
Value	Parameter value of the cell under the cursor or data value at a pseudosection point
Mode	Current mode of the model editor (drawing, selection, etc.)
Indicator	Calculation progress (forward problem, inversion, calculation of true ρa)

Section (left to right)	Contents
RMS	Misfit **RMS, %** between observed and calculated data
Messages	Stages of long calculations; after changing cell grouping, the number of data records, the number of parameters and the memory estimate

The misfit is computed as the mean relative deviation

$$RMS = \frac{100}{N}\sum_{i=1}^{N}\frac{|d_i^{obs} - d_i^{calc}|}{|d_i^{obs}| + |d_i^{calc}|}\ \%,$$

so it is robust to sign changes of the signal and to values close to zero. For a satisfactory inversion RMS is usually 2–5 %.

Mouse and keys of the main window

Action	Where	Result
Left click on a point	Data map, pseudosections	Receiver selection: it is highlighted on the map and in 3D, the **Sounding curves** window shows its curves
Mouse wheel	Pseudosection in Graphics plot mode	Move to the previous / next profile
Mouse wheel	Model editor	Neighbouring model slice
Left button + drag	3D view	Model rotation
Middle button + drag	3D view	Panning the image
Mouse wheel	3D view	Zoom
Right button	3D view	**3D settings** window
Right button	Time list	Number of neighbouring times on the map
Right button	Inversion button	Selection of Pseudo 3D / Full 3D
Right button	Slice paste button	Paste options
Right button	Plots, scales, margins around plots	Setup context menus (see “Graphics settings”)
Esc	Main window	Stop the Pseudo 3D inversion and other internal calculations (Full 3D is stopped by pressing the inversion button again)
Alt+1 / Alt+2 / Alt+3	Main window	XZ / YZ / XY slice
Model editor keys	Main window	Passed to the model editor (see “Editor keys”)

When the program is closed or a new project is opened and the project has changes, the question **Save changes to project?** is displayed (save the project: **Yes**, do not save: **No**, go back: **Cancel**).

Opening data and projects

Supported formats

A file is opened with the button or the **File / Open file** command. The file type is selected in the type list of the open dialog; it determines which reader module will read the file, so choose the type corresponding to the instrument even if the extension matches another format (for example, `*.tem` is used by TEM-FAST, Interpex, Geonics and Amira). Several files of one type can be selected in the dialog at once; they will be merged into one data set.

Type in the open dialog	Extensions	Contents, reading features
ZondTEM3D project	`*.t3d`, `*.zt3d`	Program project: data, survey layout, mesh, model, settings (see Appendix A)
ZondTEM1D field data	`*.tdf`, `*.usf`, `*.avg`	Field data in ZondTEM1D format (tdf is a text exchange format described in Appendix A), as well as USF and AVG formats. For files with repeated curves a choice of processing type is offered and the **Curve selector** window opens
TEM-FAST48	`*.tem`	TEM-FAST 48 instrument data
ERA	`*.tqq`	ERA instrument data
Podbor	`*.*`	Files of the Podbor program
Interpex	`*.tem`	Interpex format (TEMIX/IX1D)
Geonics gx7/tem	`*.gx7`, `*.tem`	Geonics PROTEM data; the **Geonics survey** window opens
Amira	`*.tem`, `*.fem`	Amira instrument data; the **Amira** window with normalization parameters opens
BEM	`*.bem`	BEM data
STEM xyz	`*.xyz`	STEM towed system (tabular xyz)
TTEM xyz	`*.xyz`	TTEM towed system (tabular xyz); the **Data units** window opens

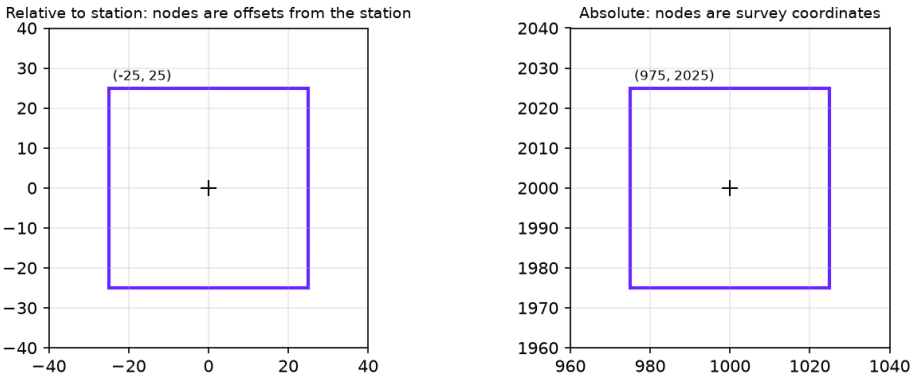
Frequency-domain data are not supported: an attempt to open such a file gives the message “Frequency domain data are not supported”.

Append data (menu **File**) adds data from another file to the current set: new stations are appended to the existing ones, after which the steps of coordinate referencing, profiles and mesh construction are repeated.

Loop geometry (menu **Options / Data**). Many formats specify loop vertices either relative to the station centre or in absolute coordinates. The reading mode is selected before opening the file:

Mode	How loop vertex coordinates are read
Auto (default)	The program compares the loop centre with the coordinate origin and with the station coordinates: if the

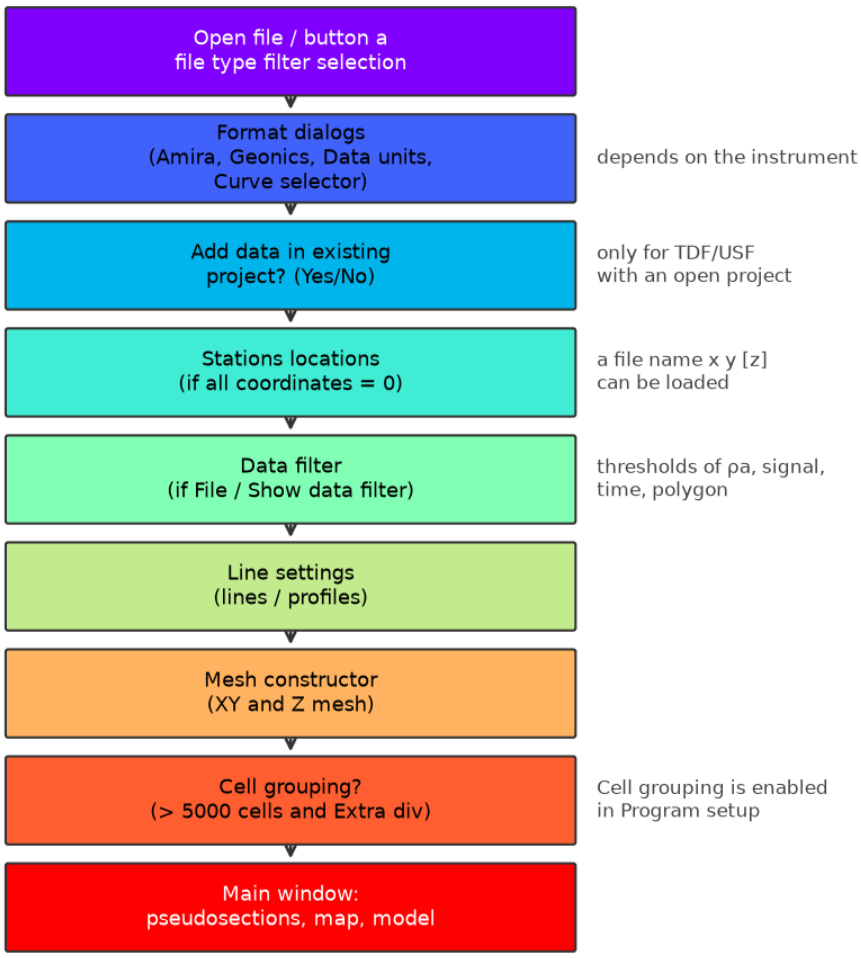
Mode	How loop vertex coordinates are read
	centre is closer to zero, the vertices are considered relative to the station, otherwise absolute
Relative to station	Vertices are always relative: station coordinates are added to them
Absolute	Vertices are always absolute



Loop geometry reading modes: left, relative coordinates; right, absolute

Sequence of dialogs on opening

When a field file is opened, the program successively displays windows in which the data are checked and completed. Windows not related to the selected format are skipped.



Sequence of dialogs when opening field data

1. Reader module dialog: specifying units, normalization and processing type (Amira, Geonics survey, Data units, processing type selection, Curve selector).
2. **Stations locations**: table of stations with coordinates. Here coordinates can be loaded from a separate file or latitude and longitude can be converted to UTM.
3. **Data filter** (if **File / Show data filter** is enabled): rejection of data by ranges, polygon and station thinning. If no data are left after filtering, the message "No data left after filtering" is displayed and the file is not loaded.
4. **Line settings**: survey profiles (lines); they are created automatically and can be edited.
5. **Mesh constructor**: construction of the three-dimensional mesh. After pressing **Apply and close** the program switches to interpretation mode.
6. **Cell grouping**: if the mesh has more than 5000 cells, the survey type is not **One line**, the automatic mesh mode is selected and intermediate nodes are specified, the program asks: "Would you like to group intermediate cells? this reduce Npar in N times". If the answer is **Yes**, intermediate cells are combined into inversion parameters (the number of parameters decreases N times), which speeds up the inversion and saves memory.

On a reading error "Error in data file" is displayed and the previous data are kept. On a project reading error, "Error in project file".

Reader module dialogs

Selection of processing type

For files in which one sounding is recorded by several curves (stacks, different currents or time ranges), the question “What kind of processing you wish to apply?” is displayed with three options:

Button	Action
Deep process	The curves of one sounding are combined into one with selection in the **Curve selector** window
Deep with shifts	The same, but with level adjustment (shift) of individual curves before combining
Averaging	All curves of a sounding are averaged without viewing

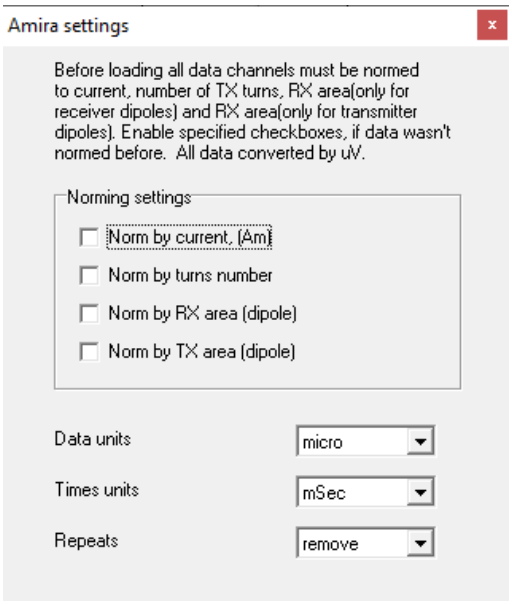
Only the vertical (Z) component is read: receiver loops and vertical magnetic dipoles (Hz induction sensors). Horizontal X and Y components, as well as electric receivers, are skipped on reading without a prompt; stations with no receivers left are not loaded. Components are not combined into a total field.

Curve selector window

The window shows all curves of the current sounding and the resulting curve (median of the checked curves). Check boxes in the legend include or exclude individual curves.

Element	Purpose
Next data point	Accept the result and go to the next sounding
Average all	Average all remaining soundings without showing the window
Shift	Curve level adjustment mode
	Smooth the resulting curve (left button); right button selects the smoothing type (**Linlog smoothing**)
Fields X, Y, Elevation, Sounding name	Coordinates, elevation and name of the sounding; they can be corrected
smooth	Smooth the resulting curve; the degree is set by the **Smoothing strength** slider
skip	Do not load the current sounding

Amira window



Amira settings window

Element	Purpose
Norm by current, (Am)	Normalize data to current if they are not normalized and the current value is in the file
Norm by turns number	Normalize to the number of turns of the transmitter loop
Norm by RX area (dipole)	Normalize to the receiver area (for coils, dipoles)
Norm by TX area (dipole)	Normalize to the transmitter area
Data units	Data units in the file (femta … unit), if they are determined incorrectly
Time units	Time units in the file (nSec, uSec, mSec, Sec)
Repeated measurements	Processing of repeated measurements: **as new channel**, as a separate receiver; **remove**, delete; **average**, average

Geonics survey window

Element	Purpose
Loop coordinates (check box)	Set the position of the transmitter loop: south-west corner (**East**, **Nord**) and side lengths (**EW side**, **SN side**)
Coil positions	Take receiver coordinates from the file
Power line frequency	Power line frequency (50 or 60 Hz); determines the instrument time set
One file one sounding	Each file is one sounding

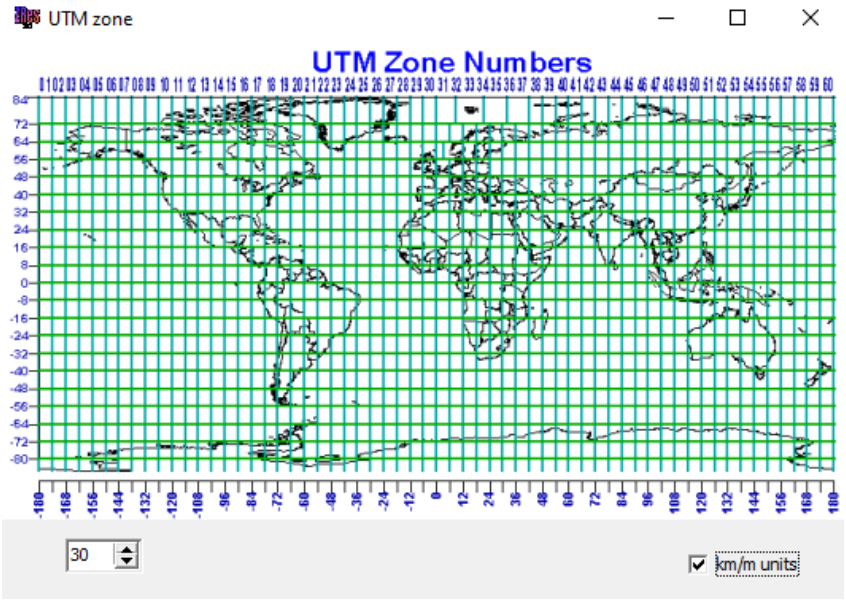
Data units window

The window is displayed for formats in which signal units are not recorded (for example, TTEM). Select the signal units (pV, nV, uV, mV, V) and mark whether the data are normalized to the

transmitter area (**/Tx(m2)**) and to current (**/A**). The program converts the data to µV/A using the loop area from the station parameters.

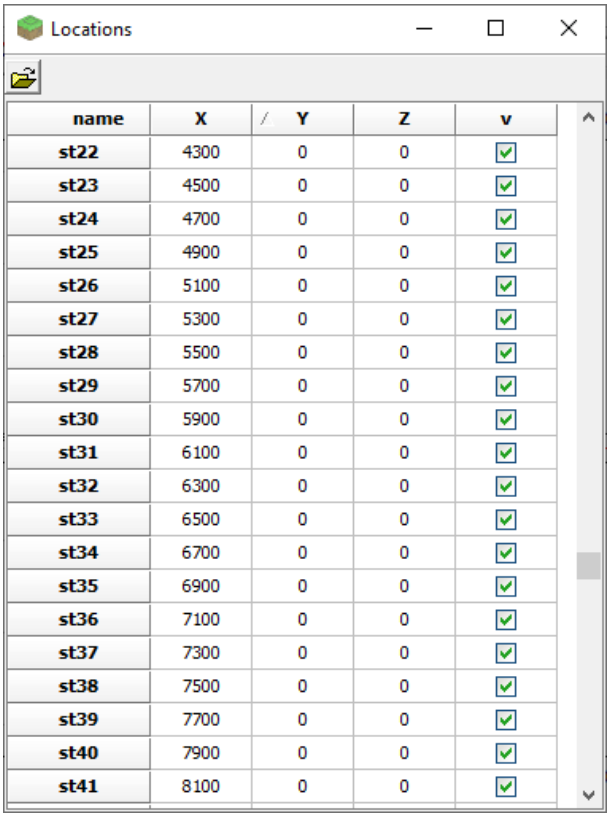
UTM zone window

The window is used when converting geographic coordinates: UTM zone number (positive for the northern hemisphere, negative for the southern), selection of a country from the list, or right click on the world map to select the zone at that point.



UTM zone window

Stations locations window



name	X	Y	Z	v
st22	4300	0	0	☑
st23	4500	0	0	☑
st24	4700	0	0	☑
st25	4900	0	0	☑
st26	5100	0	0	☑
st27	5300	0	0	☑
st28	5500	0	0	☑
st29	5700	0	0	☑
st30	5900	0	0	☑
st31	6100	0	0	☑
st32	6300	0	0	☑
st33	6500	0	0	☑
st34	6700	0	0	☑
st35	6900	0	0	☑
st36	7100	0	0	☑
st37	7300	0	0	☑
st38	7500	0	0	☑
st39	7700	0	0	☑
st40	7900	0	0	☑
st41	8100	0	0	☑

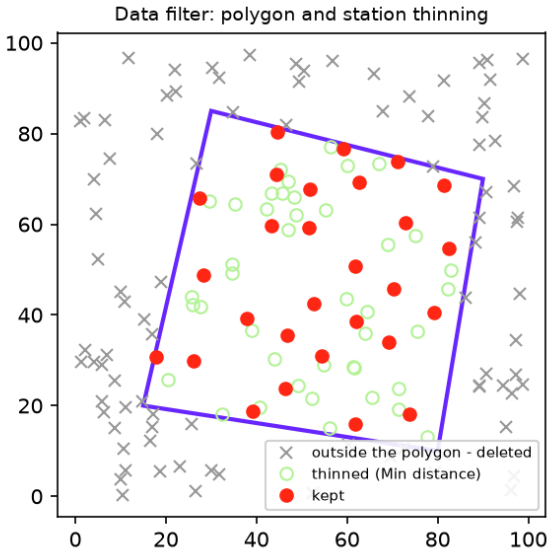
Stations locations window

The table contains one row per station: name, X, Y, Z. Coordinates can be entered manually, pasted from Excel (**Ctrl+V**) or loaded from a file.

Element	Purpose
	Load station coordinates from a text file (format: name, X, Y and optional Z; see Appendix A)
LL->XY	Convert longitude (column X) and latitude (column Y) to UTM metres
Right click on an X or Y cell	Enter an increment: starting from the next row, the column values are filled as previous + increment (convenient for a profile with a constant step)
Ctrl+C / Ctrl+V	Copying and pasting selected cells

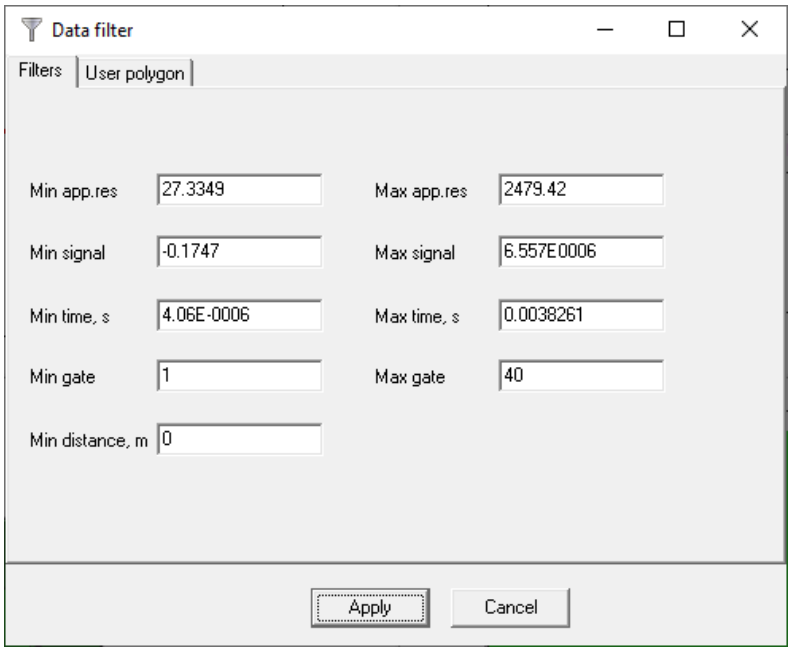
Rows in which coordinates are undefined are highlighted in colour. Stations with identical coordinates will coincide on the map; check such rows before building the mesh.

Data filter window



Data filter: polygon and station thinning

The window opens when a file is loaded if **File / Show data filter** is checked. On the **Filters** tab ranges are set: values outside them are removed (marked as missing).

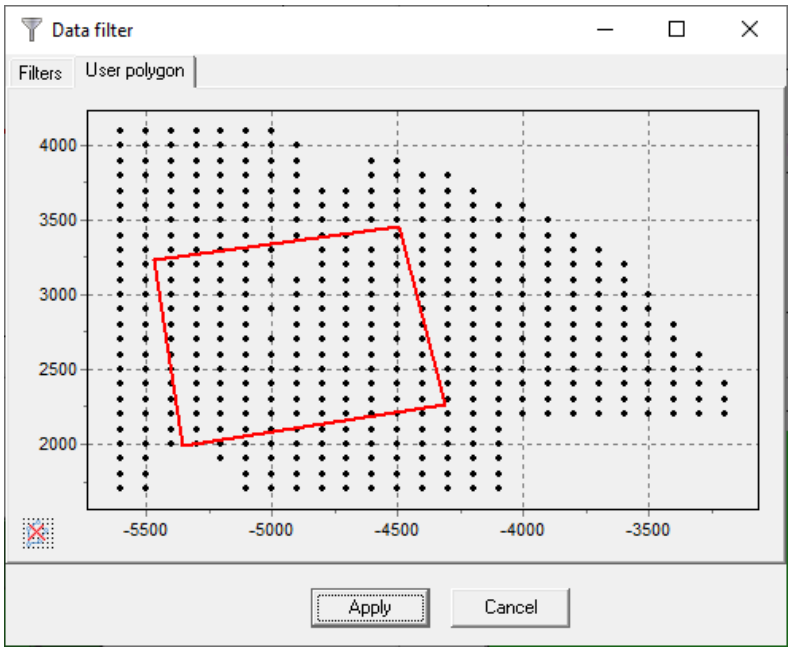


Data filter window, Filters tab

Field	What it limits
Min app.res / Max app.res	Late-time apparent resistivity, Ohm·m
Min signal / Max signal	Normalized signal, μV/A (absolute value)
Min time, s / Max time, s	Time range, s
Min gate / Max gate	Range of gate (channel) numbers
Min distance, m	Station thinning: stations located no closer than the specified distance to each other are kept (0 means no thinning)

Thinning is performed by the “farthest point” rule: the station farthest from the survey centre is taken first, then the station farthest from those already selected, and so on until the distance becomes less than the specified one. Therefore the remaining stations are distributed over the area as uniformly as possible.

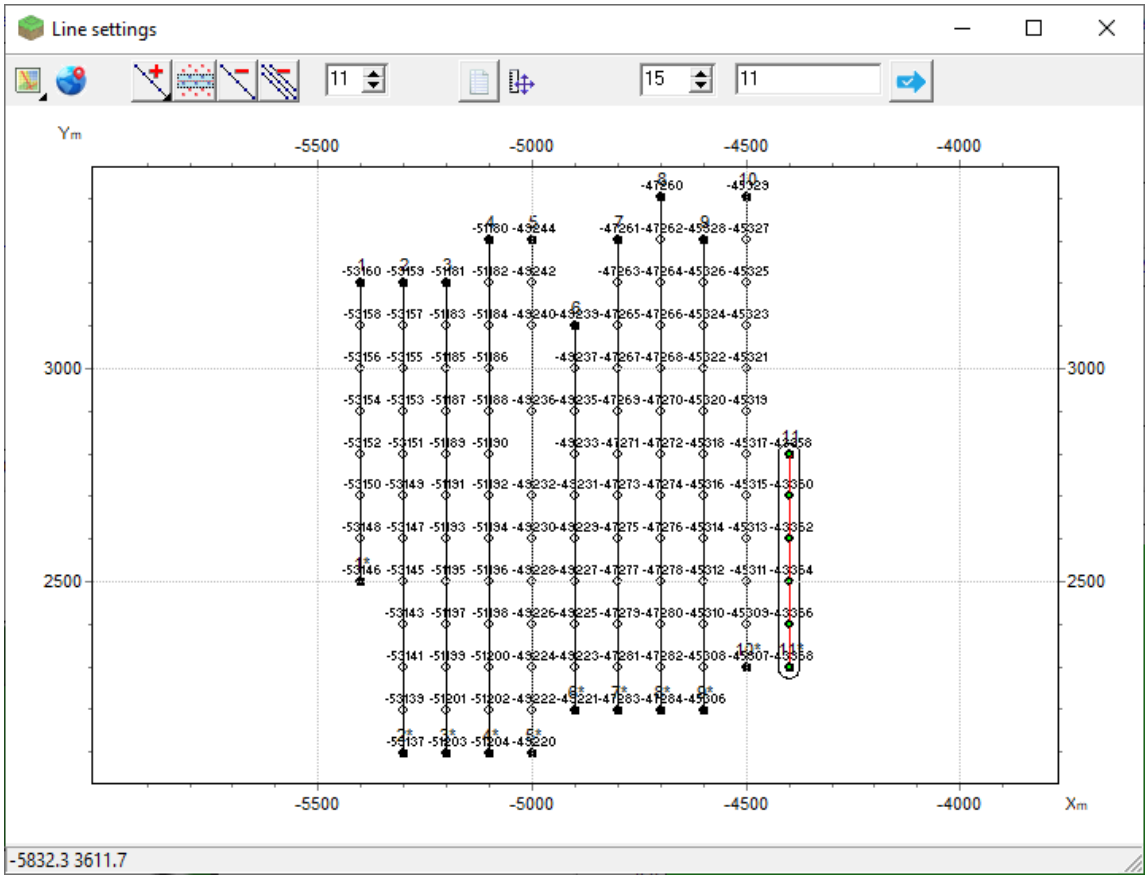
On the **User polygon** tab a polygon is drawn on the station plan: left click adds a vertex, right click closes the polygon. Stations outside the polygon are removed. The **Remove polygon** button deletes the polygon. **Apply** applies the filter, **Cancel** loads the data without filtering.



Data filter window, User polygon tab

Line settings window

Profiles (lines) are needed for building pseudosections and plots and for operations along a line (Rx-Tx alignment, saving line data). When a file is opened, profiles are built automatically; the window allows them to be checked and corrected. Later the window is opened with the **Options / Set survey lines** command.



Line settings window

Button	Purpose

Button	Purpose
	Load a satellite map as a background (coordinates must be in UTM)
Map	Left button opens the background menu (**Map**, raster map; **Topography**, relief); right button hides the background
Add new line	Draw a new line: left button adds a point, right button finishes the line. Right click on the button itself opens the line menu
Activate point in selection rectangle	Include in the active line the stations falling into a band around the line (band width is **Width of the selection rectangle, %**)
Remove active line / Remove all lines	Delete the active line / all lines
Active line index, Line name	Number and name of the active line
Set coordinates in table	Coordinates of line points in a table
Plot scale	Scale 1:1 or fit to window
Goto Interpretation mode	Close the window and proceed to mesh construction

Line menu (right click on the **Add new line** button):

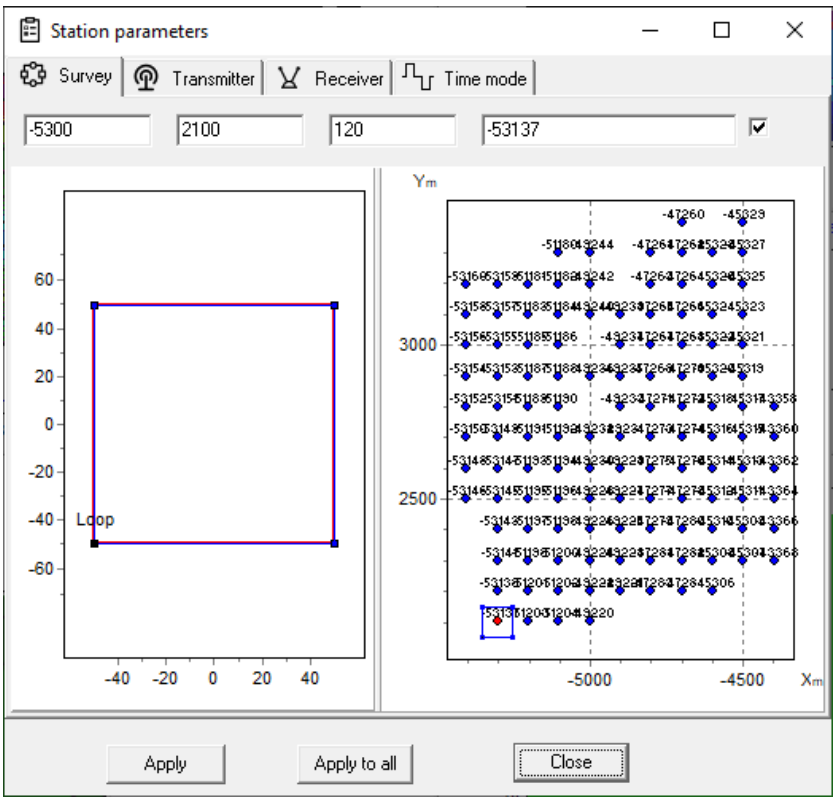
Item	Purpose
Set line coordinates	Enter the coordinates of line vertices
Cursor to near point	While drawing, “snap” vertices to the nearest station
Invert line	Reverse the line direction (start ↔ end)
Create lines auto	Build lines automatically; parameters are set in the **Settings** window: maximum turning angle between neighbouring segments (30° by default), allowed growth of the mean deviation of points from a straight line when merging fragments, %, and the neighbour search method (**Auto**, preferred direction **SN**, **WE**, **NW**, **NE** or **Parallel lines (auto)**)

Mouse and keys in the window:

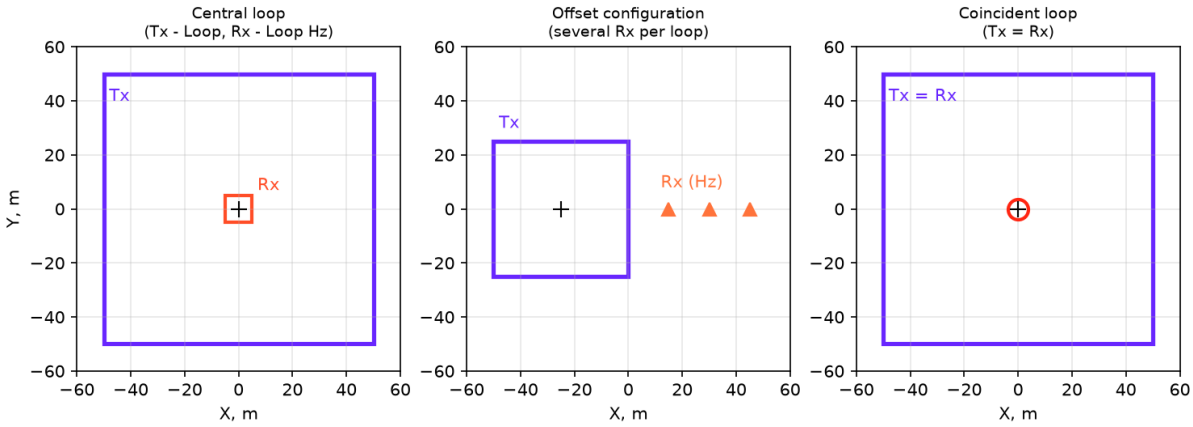
Action	Result
Left click on a station	Include or exclude the station from the active line
Right click on a station	Enter the X, Y, Z coordinates of the station
Ctrl + click on a station	Highlight the receivers associated with the same transmitter loop
Hold **Alt**	Brush mode: a circle around the cursor; the left button (click or drag) includes stations in the line, the right button excludes them; the mouse wheel changes the brush radius

Station parameters window

The window is opened with the button and shows the parameters of the station selected on the map (or the first station). Changes take effect after the **Apply** button (current station) or **Apply to all** (changes of the current tab are transferred to all stations with the same original geometry). **Close** closes the window without applying.



Station parameters window, Survey tab

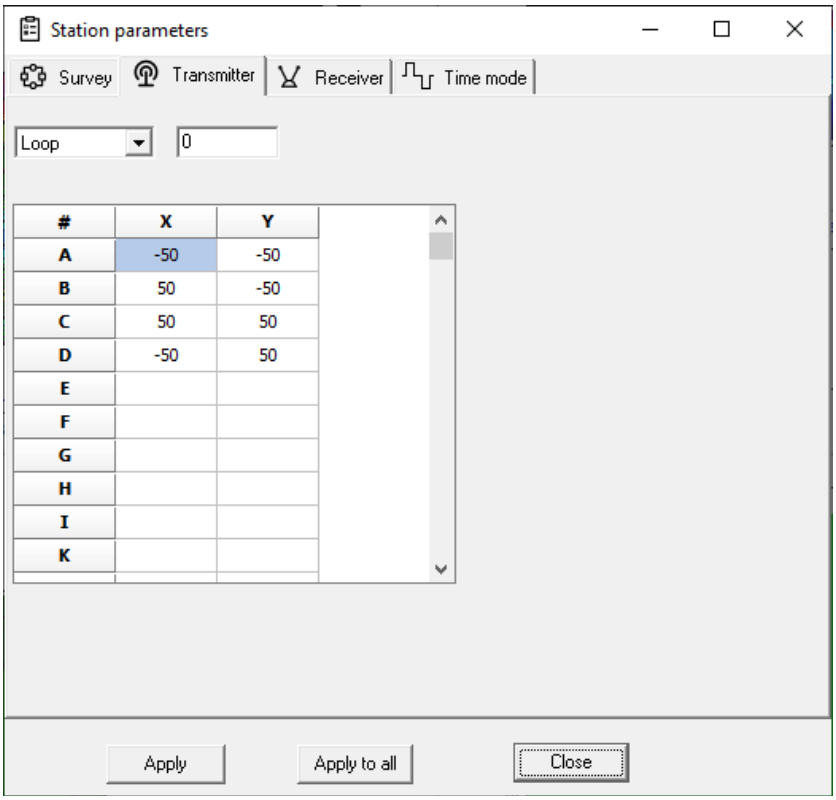


Typical TEM configurations: transmitter loop Tx and receiver Rx

Survey tab

Station plan (transmitter, receivers) and the fields **X**, **Y**, **Z** (station coordinates), **Sounding title** (name). The “local coordinates” check box means that loop vertices are specified relative to the station X, Y.

Transmitter tab



Station parameters window, Transmitter tab

Element	Purpose
Transmitter type	**VMD** (vertical magnetic dipole, a small loop) or **Loop** (loop of arbitrary shape)
Vertex table	Loop vertex coordinates (the loop is closed automatically). **Ctrl+C / Ctrl+V** is supported
Elevation	Transmitter elevation above the surface, m (0 for ground loops)

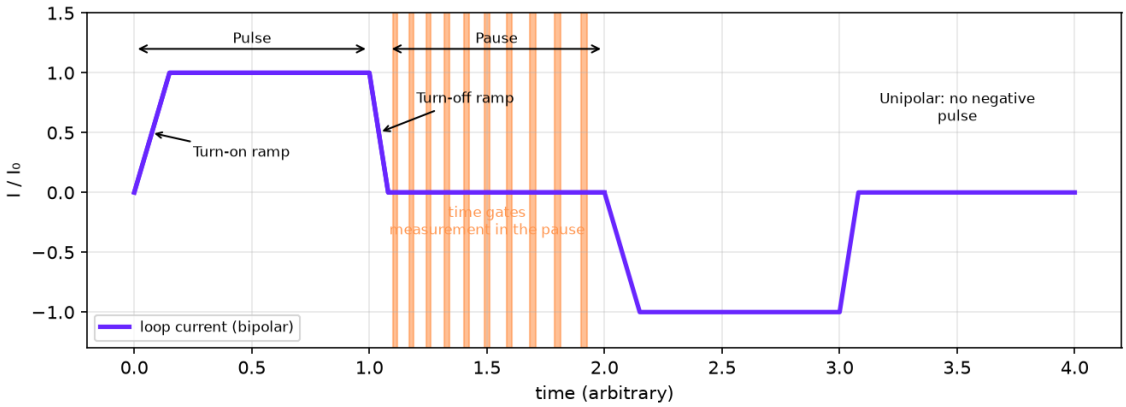
Receiver tab

Element	Purpose
Receiver type	**Loop Hz** (receiver loop) or **Hz** (point sensor of the vertical component; the effective area is specified in the data)
Receiver number	If the station has several receivers, select the active one
Table	Receiver loop vertices or point coordinates
Elevation	Receiver elevation above the surface, m
Factor	Multiplier of the calculated response; −1 if the sign of the instrument response is opposite
Same center	All point receivers of the station have a common centre: moving the centre of one moves all of them
dF/dt	On: dB/dt (EMF) is measured; off: magnetic field B

Context menu of the tables (right button):

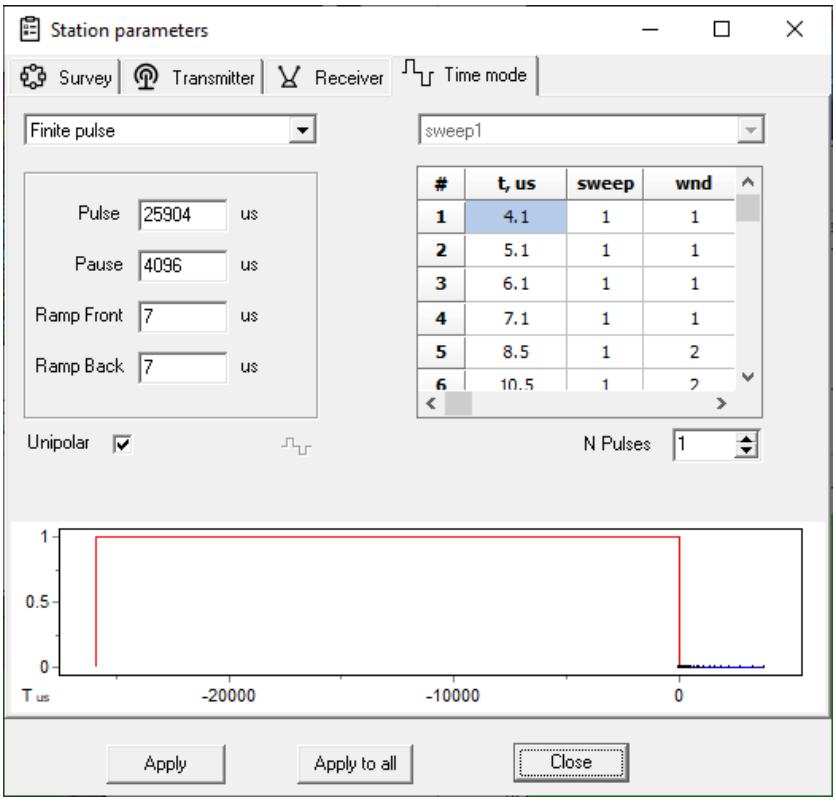
Item	Purpose
Create square loop	Create a square loop with the given side (m) around the centre
Turn over	Reverse the order of vertices (changes the sign of the loop response)
Get size	Show the loop area
Replace with loop	Replace the point Hz receiver with a square loop centred at the receiver point
Change sign	Change the sign of the observed signal of the active receiver

Time mode tab



Current waveform: pulse, pause and ramp durations

Element	Purpose
Sweep index	Series number (if the instrument has several ranges or repetition frequencies)
Pulse type	**Step**: step (instantaneous turn-off, infinitely long pulse; suitable if the pulse is much longer than the pause and the ramps are small); **Finite pulse**: finite pulse taking into account duration, pause and ramps; **User defined**: arbitrary current waveform from a table
Pulse length, us	Pulse duration, μs. Affects late times
Pause length, us	Pause duration, μs
Turn-off ramp, us	Turn-off ramp duration, μs. Affects early times. A negative value means times are counted from the end of the ramp, a positive value from its beginning
Turn-on ramp, us	Turn-on ramp duration, μs
Number of pulses	How many preceding pulses to take into account (1–2 is usually enough)
Unipolar	The instrument uses unipolar pulses
Import pulse shape	Open the **Pulse shape** window



Station parameters window, Time mode tab

Pulse shape window

The "time–amplitude" table describes one pulse of arbitrary shape. Buttons: **+** adds a node, **−** deletes a node, **x** inserts intermediate nodes, load the shape from a file, save the shape to a file (format in Appendix A). **Ctrl+C / Ctrl+V** exchanges with Excel.

Workflow: loading field data

1. Select the **Options / Data / Loop geometry** mode (usually **Auto**) and enable **File / Show data filter** if rejection is needed.
2. Press , select the instrument file type and the files.
3. Answer the questions of the reader module (units, normalization, processing type).
4. In **Stations locations** check the coordinates; if necessary, load them from a file or convert LL→XY.
5. In **Data filter** discard obviously bad times and channels; for a dense survey set **Min distance**.
6. In **Line settings** check the automatic profiles, rebuild them if necessary (**Create lines auto**) and press **Goto Interpretation mode**.
7. In **Mesh constructor** define the mesh and press **Apply and close**; accept cell grouping for a large survey.
8. Check the station parameters (): loop shape and elevation, current waveform, signal sign. Save the project (`*.t3d`).

Viewing and processing data

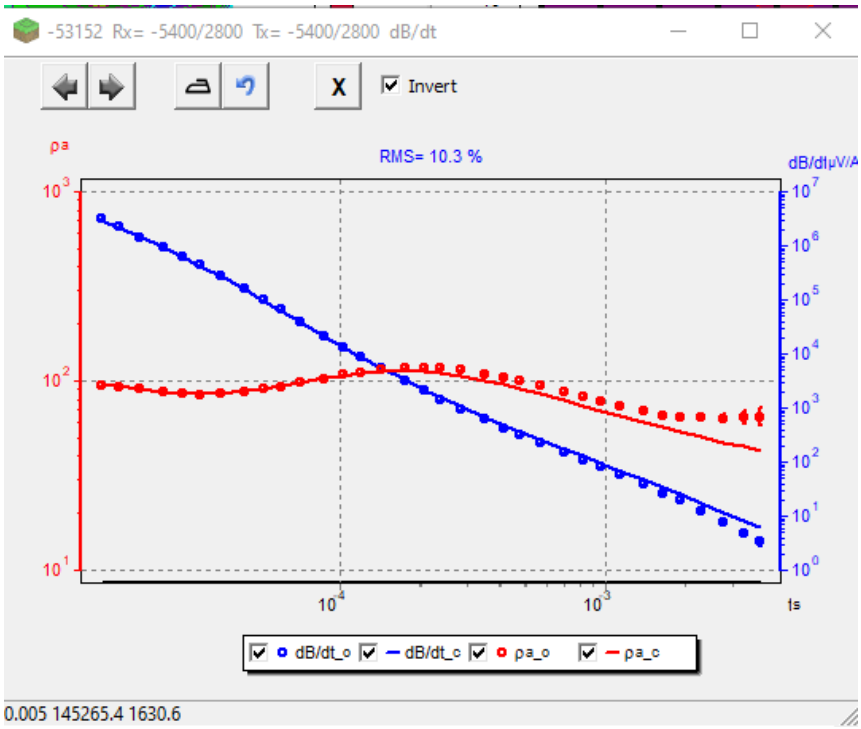
Data display modes

The data display mode is selected in the **Options / Data** menu. Display settings (scales, palettes, levels) are stored separately for each combination of display mode and data type, so the last used ones are restored when switching.

Menu item	What it changes
Pseudosection	The lower windows show pseudosections along the active profile: horizontally, distance along the profile; vertically, time (logarithmic scale)
Graphics plot	The lower windows show plots: each curve is one time, horizontally the position on the profile. The mouse wheel switches the profile
Sounding curves	Show or hide the sounding curves window of the selected receiver (same as the  button)
Contour map / Point map	Data map at the selected time: contours or coloured points
Observed data / Calculated data / Data misfit	What the second pseudosection shows: observed data, calculated data or misfit
Apparent resistivity / Signal	Display apparent resistivity (Ohm·m) or normalized signal (μV/A)
Error gates	Show measurement errors (error gates) on plots
True apparent resistivity	Show the “true” apparent resistivity of a homogeneous half-space instead of the late-time asymptote. Calculation of the tables takes some time (progress is shown in the status bar). Used for display only
Smooth data	Window for smoothing observed data
Loop geometry	Method of reading loop coordinates at the next file opening (Auto / Relative to station / Absolute)

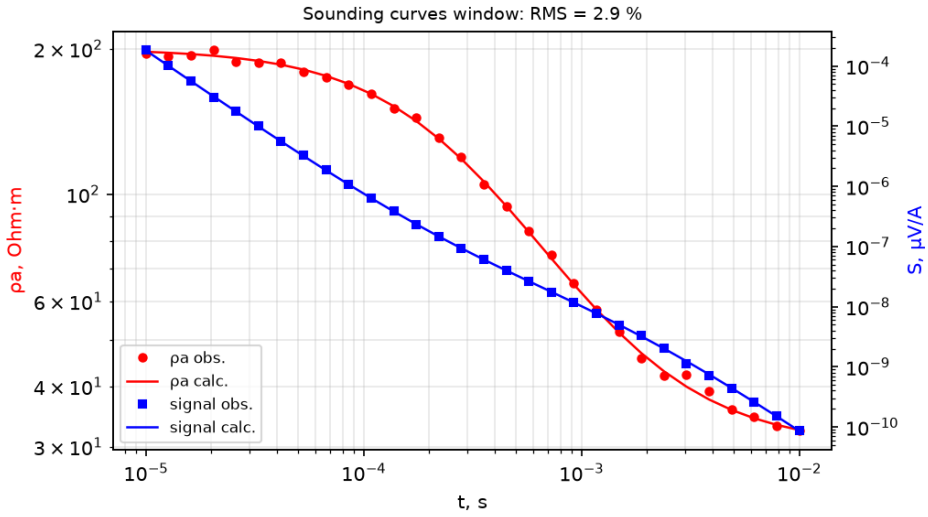
> **Tip.** The late-time apparent resistivity is strongly distorted at early times (especially for small loops and resistive sections) and may produce a “false” conductive upper layer. To assess data quality it is more convenient to look at the signal, and for geological reading, at the true ρa.

Sounding curves window



Sounding curves window

The window shows the observed and calculated curves of the selected receiver on a log-log scale and the RMS misfit for this receiver. The receiver is selected by clicking a point on the map or pseudosection or with the arrow keys in the window.



Sounding curve: dots are observed data, the line is the calculation

Button / element	Purpose
◀ ▶	Previous / next record (same as the ← / → keys)
	Smooth the curve: left button, the current one; right button, all curves
	Undo the last change
X (deletion mode)	The cursor becomes a circle; a left click deletes all points inside the circle. The mouse wheel changes the

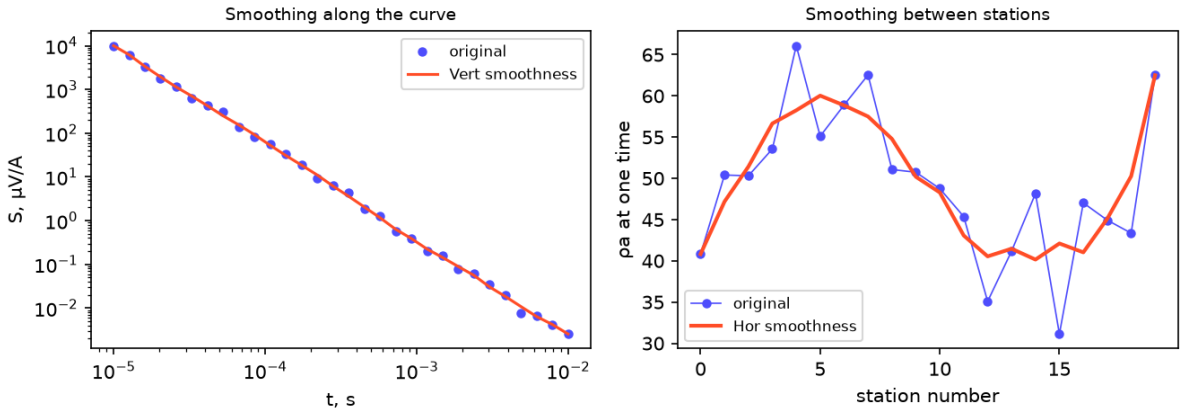
Button / element	Purpose
	circle radius (4–200 pixels)
Invert	If the check box is cleared, the station data are not included in the inversion misfit (they are shown but not fitted); the model beneath the station is formed by its neighbours through smoothing. If all stations are excluded, the message “All stations are excluded from the inversion” is displayed at start
Status bar	Time and value under the cursor

Mouse in the window:

Action	Result
Dragging a point with the left button	Change the point value
Right click on a point	Point menu: **Delete point**, delete; **Smooth point**, replace the value with one smoothed over neighbours; **Origin point**, restore the original value; **Delete all**, delete all points of the curve
Shift + right button	Do not show the point menu
Mouse wheel (in X mode)	Radius of the deletion circle

Deleted points do not participate in the inversion and in the RMS calculation; they are not shown on plots. Records with all points deleted can be removed from the project with the **Options / Extra / Remove masked records** command.

Data smoothing



Smoothing in time (left) and between stations (right)

The **Smoothing settings** window is opened with the command **Options / Data / Smooth data**.

Element	Purpose
Time slider	Degree of smoothing of each curve along the time axis
ρa check box	Smooth apparent resistivity instead of the signal (useful for curves with a large dynamic range)
1D / 3D	1D: each station is smoothed separately; 3D: neighbouring stations are additionally taken into account by coordinates

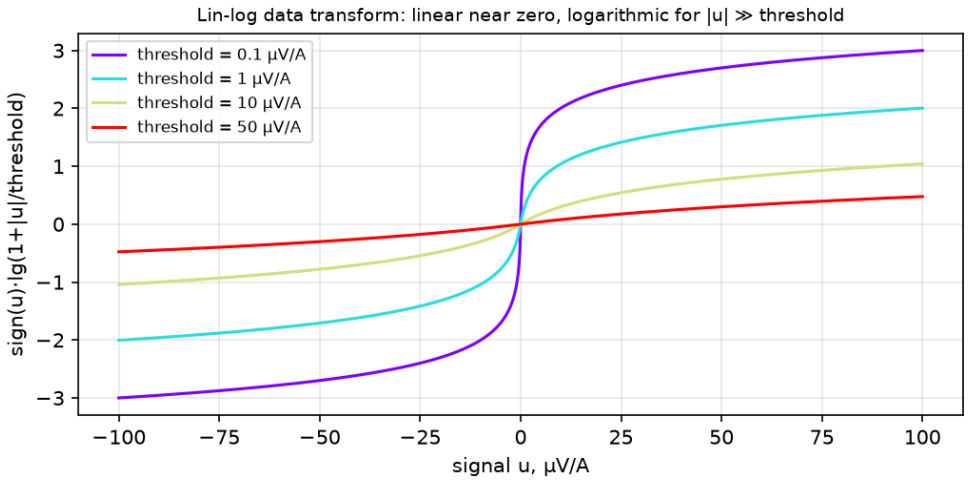
Element	Purpose
Slider between stations	Degree of smoothing between neighbouring stations (0 means off)
set positive	Make all smoothed signal values positive
segments	Smooth each time segment (series) separately, for instruments with several ranges
Preview	Smooth the data with the current settings and show the result on the plots. If the window is closed without **Apply**, the original data are restored
Apply	Save the smoothed data as observed and close the window

> **Attention.** Smoothing changes the observed data of the project. If the original data must be kept, save the project under another name before smoothing.

Inversion data norm (Lin-log threshold)

The TEM signal varies by several orders of magnitude and may change sign. Therefore the inversion compares not the values themselves but their transform. The norm type is set with the **Options / Inversion / Resolution / Data norm** command (the item shows the current mode), in the input window:

Input	Mode	Description
`*` or `-1`	**linlog_auto**	Lin-log with automatic threshold: the threshold is chosen from the data distribution
`0`	**log**	Logarithm of the late-time apparent resistivity. If the data contain negative or zero values, linlog_auto is used automatically
number > 0	**linlog_fixed**	Lin-log with a given threshold, μV/A. A value close to the noise level is recommended



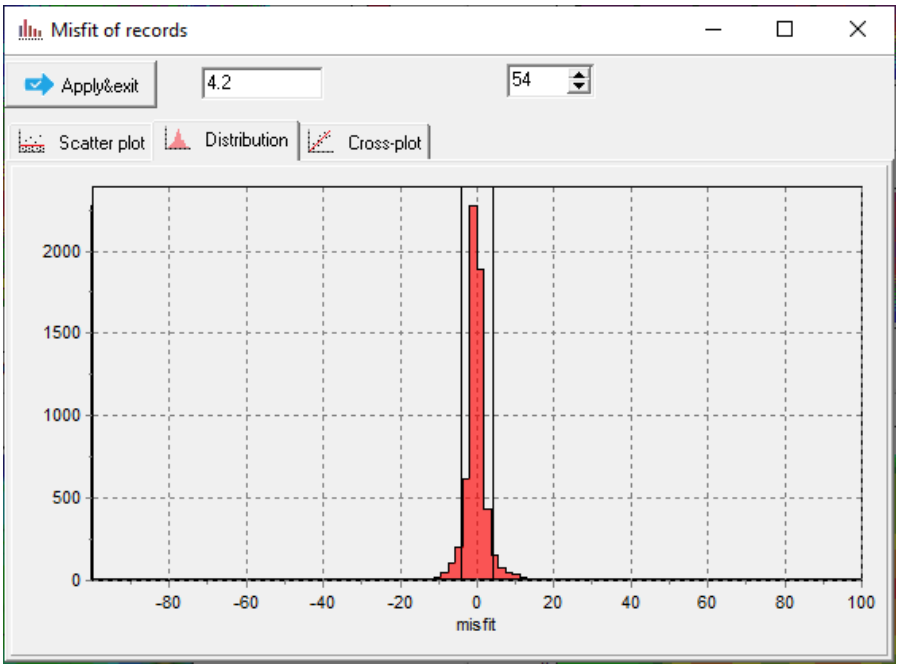
Lin-log transform: linear scale near zero, logarithmic at large values

The lin-log transform behaves as a logarithm for signals much larger than the threshold and linearly for signals near zero. Therefore noise values at late times (close to zero and of different sign) are not “inflated” by the logarithm and do not receive excessive weight.

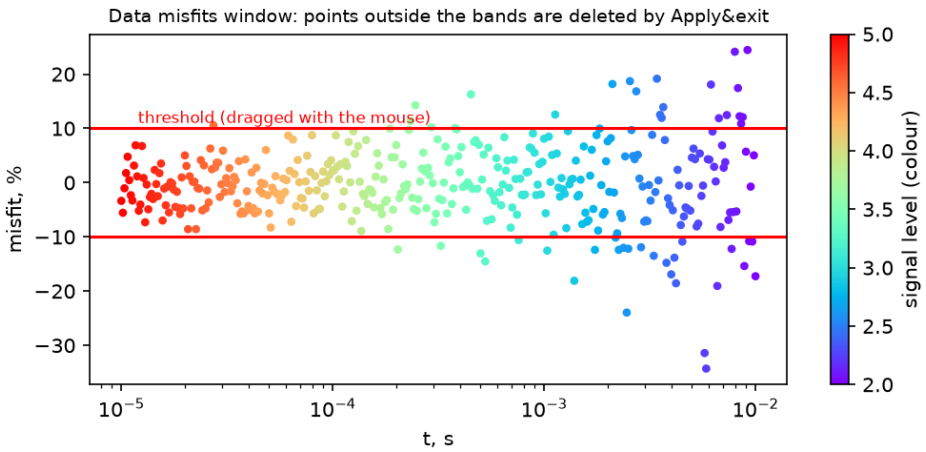
Removing data with large misfit

The **Options / Extra / Remove data with big misfit** command is available after the forward problem or inversion has been calculated. The **Misfit of records** window contains:

- a plot of misfit versus time: each point is one measurement, the colour is the signal magnitude (blue, weak; red, strong);
- a cross-plot of observed and calculated values;
- a histogram of the misfit distribution (the number of bins is set in the adjacent field).



Misfit of records window

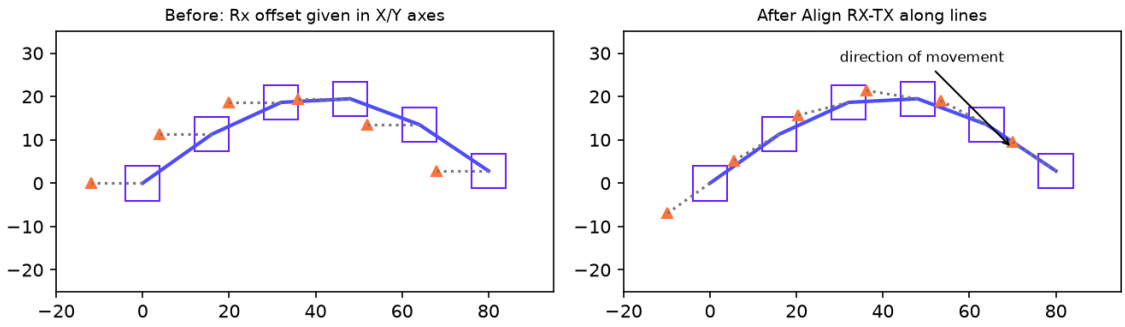


Rejection scheme: points outside the threshold band are removed with the Apply&exit button

The initial threshold equals twice the current RMS. The threshold (%) is entered in the field or set by dragging the horizontal lines on the plot. **Apply&exit** removes records with misfit above the threshold and closes the window.

Rx-Tx alignment along lines

The **Options / Extra / Align RX-TX along lines** command is intended for offset configurations (loops or dipoles) moved by different operators and for towed systems. The Rx offset relative to Tx in the file is considered to be given in the instrument frame (X axis is the direction of motion) and is rotated by the profile azimuth at each station.



Rx-Tx alignment: the offset is rotated along the profile direction

Before execution the program asks to confirm that the lines are oriented in the direction of motion. Automatically built lines point east (north for meridional lines); if the motion was in the opposite direction, reverse the line with the **Invert line** command in the **Line settings** window. Without profiles, “No lines. Define the lines first.” is displayed. After execution the number of aligned stations and the number of stations aligned earlier (they are not rotated again) are reported.

Other data operations

Command	Purpose
Options / Extra / Remove masked records	Delete receivers with all points deleted and stations without receivers
Options / Extra / Copy calc to measured	Copy calculated data to observed (create a synthetic data set for testing the inversion)
Options / Import/Export / Save calculated data	Save calculated data in tdf format: of the current station, the current line or all data (the type is selected in the save dialog)
File / Edit file	Open the original data file in Notepad

Workflow: data quality control

1. Switch to **Signal** and **Pseudosection**; look through all profiles (line list or mouse wheel in plot mode).
2. Find stations with anomalous behaviour (vertical bands, sign changes) and open their curves by clicking a point.
3. In the **Sounding curves** window delete outliers (**X** mode or right click on a point) and, if necessary, exclude the station from the inversion (**Invert** check box).
4. Check the configuration parameters of such stations (): area, elevation, sign (**Factor**, **Change sign**).

5. If noise is noticeable, apply **Smooth data** with a moderate degree; compare the result with the **Preview** button.
6. Select the data norm (usually linlog_auto) and save the project.

Main menu

This chapter lists all items of the main menu with a brief description. A detailed description of the windows opened from the menu is given in the corresponding chapters (references are given in the tables). Items next to which a check mark is shown in the menu are toggles.

File menu

Item	Purpose
Open file	Open a data or project file (*.t3d). Same as the  button. The sequence of dialogs on opening is described in the chapter “Opening data and projects”
Append data	Add stations from one more data file to the current set. The format and configuration parameters must be compatible with the already loaded data; the mesh and model are rebuilt after adding
Show data filter	Toggle: show the **Data filter** window when loading a data file
Save file	Save the project (*.t3d) or export data/model. Same as the 
Edit file	Open the original data file in Notepad
Print preview	Preview and print the current model section
Reg file	License registration file dialog
Registration	Service registration function (developer password required)
Recent	List of recently opened files
Exit	Close the program. If the project has been changed, the program offers to save the changes

Options menu

General items

Item	Purpose
Project information	Project information (title, area, organization, client, dates, equipment, operators). Saved in the project
Mesh constructor	Model mesh construction window (chapter “Mesh constructor”)
Program setup	Inversion parameters and inversion mesh parameters (chapter “Inversion”). Same as the 
Set survey lines	The **Line settings** window: survey profiles along which pseudosections and plots are built

Inversion submenu

Item	Purpose
Set apriori boundaries	Set a priori boundaries: polygons on horizontal nodal planes of the mesh. In Full 3D inversion vertical smoothing across the marked part of the plane is weakened 100 times. Cancelling the dialog deletes all boundaries; rebuilding the mesh and opening a project also reset them (boundaries are not written to the project)
Show progress	Show the **RMS vs iteration** window during the inversion
Optimization / Line search	Selection of the smoothing parameter at each iteration. Usually speeds up convergence in terms of the number of iterations, but each iteration takes 4–6 times longer; stopping in a local minimum is possible
Optimization / Dec factor	Available when **Line search** is off: the smoothing parameter is divided by this value (1–3, 1.1 by default) at each iteration
Optimization / Lim based inv	Inversion with limits taken into account: if cell limits are set in a narrow range, cells that go beyond the limits receive less weight, and the remaining cells are fitted better
Optimization / Layered term	Additional “layered” term of the Full 3D inversion objective function: a penalty for weak correlation of the vertical resistivity profiles of neighbouring cell columns (**Extra layered term** dialog). Settings are saved in the project
Resolution / none, x 1, x 2, x 4	Sensitivity normalization: reduces the influence of cells with high sensitivity (near the surface, near loops) and lets cells with low sensitivity (at depth) change more. x 1 by default
Resolution / Data norm	Inversion data norm: lin-log with automatic threshold, logarithm of ρa or lin-log with a given threshold (section “Inversion data norm”)
Smoothness / Median	Smoothing reference model m0 is the median of the current model (default). A small Smoothing factor (0.001–0.1) gives a better fit, a large one (0.1–1) a smoother model. The best choice for Focused
Smoothness / Start model	m0 is the starting (a priori) model from the editor, for example a previous or imported result. Smoothing factor controls the closeness to m0
Smoothness / Previous	m0 is the model of the previous iteration. With Occam it gives a good fit and smooth convergence; a large Smoothing factor (0.2–1) is stable but slower
Smoothness / Norm / L0, L1, L2	Norm of the smoothing term: L0, compact bodies with sharp boundaries; L1, blocky model; L2, smooth model (classical least squares)

Item	Purpose
Smoothness / Diagonal filter	Take diagonal neighbours into account in the smoothing operator (for dipping structures)
Smoothness / Robust	Robust smoothing: blurs sharp contrasts and local bodies less
Smoothness / Stabilizer	Stabilizer of the smoothing term (0.01–1, 0.5 by default)

Data submenu

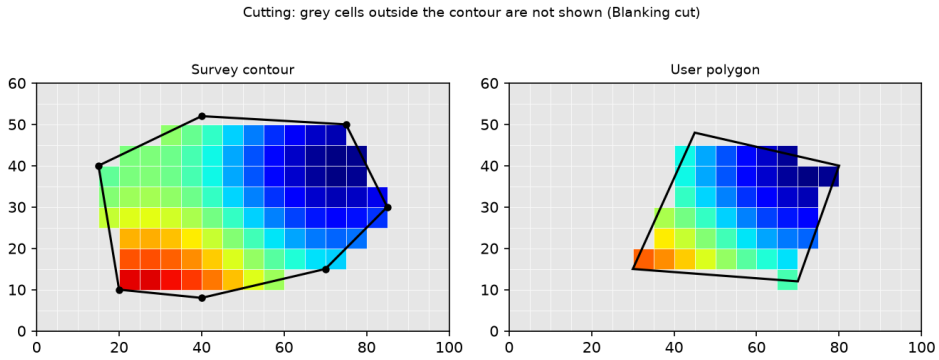
Described in the chapter “Viewing and processing data” (section “Data display modes”).

Model submenu

Item	Purpose
Block-section / Smooth section	Show the model section as blocks (cells) or as a smoothed (interpolated) image
2D mode	2D mode toggle for a single-profile survey (switched on manually). An edit of the current section is copied to all parallel sections; if the mesh is built with the **One line** survey type, when the mesh is applied the cells across the profile are combined into one inversion parameter, and the model remains homogeneous across the profile
Plane / XZ, YZ, XY	Model section direction: vertical along X (**Alt+1**), along Y (**Alt+2**), horizontal (**Alt+3**)
Plane / Autosize	Fit the section axes to the window size
Model histogram	Histograms of the resistivity distribution of the model and data
Colorscale auto	Automatically update the model colour scale range after each iteration
Copy colorscale	Transfer the model colour scale to the pseudosections and the data map
Set 1D model	Set a layered 1D model (values and limits versus depth) for all cells

Cutting submenu

Item	Purpose
Survey contour	Cut (blank) cells outside the survey outline
User polygon	Cut cells outside a user polygon (the polygon is selected in a dialog)
Groups cutted off cells	Combine cut cells into 1D columns in the inversion (reduces the number of parameters)
Blanking cut	Do not draw cut cells on the section



Model cutting by the survey outline

Import/Export submenu

Item	Purpose
Background 2D / Load background, Remove current	Background image (bmp, png, sec, sgy) for the current XZ/YZ section; delete the background of the current section
Background 3D / Load background	XYZ background for the 3D view
Background 3D / Load satellite map	Satellite map for the 3D view (map type is set in **Help / Map type**)
Background 3D / Remove background	Delete the 3D background
Save calculated data	Calculated data in tdf format: current station, current line or all data
Build-in mod3d	Import a MOD3D model and embed it into the current mesh
Export mod3d	Export the model to MOD3D
Build-in mod2d's	Interpolation of points of the selected MOD2D files into the whole 3D mesh
Export mod2d	Export the current section to MOD2D
Plot MOD2D model	Show a MOD2D or LAY model on the current section
Create video	Record an AVI video of the 3D view
Save rotations	Save 11 images of the 3D view with rotation by 18°
Save slides	Save all sections along the current direction as images
Save 3D image	Save a 3D image with the specified resolution
Output settings	Scale and resolution of output images
Export to Paraview	3D scene to a VTK file for ParaView
Export to Oasis	3D scene to 3D DXF for Oasis montaj
Draw to surfer	Horizontal model slices at specified depths for Surfer
Draw to voxler	Model to a Voxler data file (voxout.dat in the program folder)

Boreholes submenu

Item	Purpose

Item	Purpose
Create/Edit borehole data	Creating and editing borehole data (only in XZ and YZ sections)
Load borehole data	Load a log/lithology file (crt) or a mod1d model
Remove borehole data	Delete borehole data from the project
Set column's width	Borehole column width, % of the image size (0.05–5)
Edit positions	Editing borehole coordinates

Extra submenu

Item	Purpose
Remove data with big misfit	Delete records with misfit above the threshold (after calculation)
Remove masked records	Delete receivers with completely deleted points and stations without receivers
Copy calc to measured	Calculated data → observed (synthetics). The item is visible only when observed data are not loaded
Align RX-TX along lines	Rotation of the Rx-Tx offset along the profile direction

Buffer menu

The buffer stores up to four models for comparing inversion results with different settings.

Item	Purpose
Model 1 … Model 4	Empty slot: save the current model. Filled slot: the **Working with buffer** dialog; **OK** replaces the model in the slot with the current one, **Cancel** loads the model from the slot
Open	Show all buffer models in one window

Help menu

Item	Purpose
About	Version and program information
Manual	Open the manual (ZondTEM3D_eng.pdf)
Server	Update and news server: zond-geo.com, zond-geo.co, kaminae.narod.ru
Check for updates	Check for a new version
Restart library	Restart the computational library. If the forward problem or inversion stops every time, check the antivirus settings: it may block the computational module
Map type	Satellite map type: Sat1, Sat2, Sat3, Streets, Topo
Show news	Show news at startup
Send message to us	Send a message to the developers

Item	Purpose
License info	License information

Inversion button pop-up menu

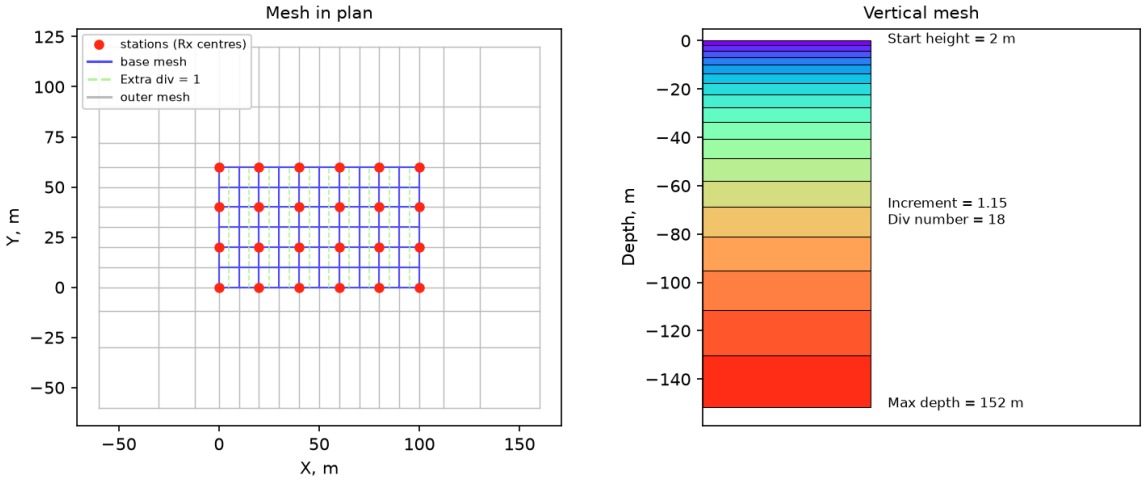
A right click on the inversion button  opens the inversion type menu:

Item	Purpose
Pseudo 3D inversion	Fast quasi-3D inversion: 1D inversion beneath each station with smoothing between neighbours (chapter “Inversion”)
Full 3D inversion	Full 3D inversion with calculation of the forward problem and sensitivities by the finite element method in the external 64-bit module

Mesh constructor

Purpose

The model mesh is a three-dimensional system of cells in which resistivity is specified. It is used both for the forward problem (finite element method) and for the inversion. The **Mesh constructor** window opens automatically after loading a new data file, and also with the **Options / Mesh constructor** command.



Mesh in plan (left) and vertical mesh (right)

The mesh is built in two stages:

1. **Mesh in plan (XY).** Nodes of the base mesh are tied to the receiver centres (stations). The main mesh directions are chosen according to the profile direction. Intermediate nodes (**Extra div**) may be added between base nodes.
2. **Vertical mesh (Z).** The thickness of the first layer (**Start height**) and the increase factor (**Increment**) are chosen automatically from the range of depths to which the data are sensitive; topography is transferred to the underlying layers with attenuation (**Topo coeff**).

When the forward problem is calculated, the program automatically adds outer cells to the model mesh (horizontal and downward extension) required to satisfy the boundary conditions. These cells are not shown and are not fitted by the inversion.

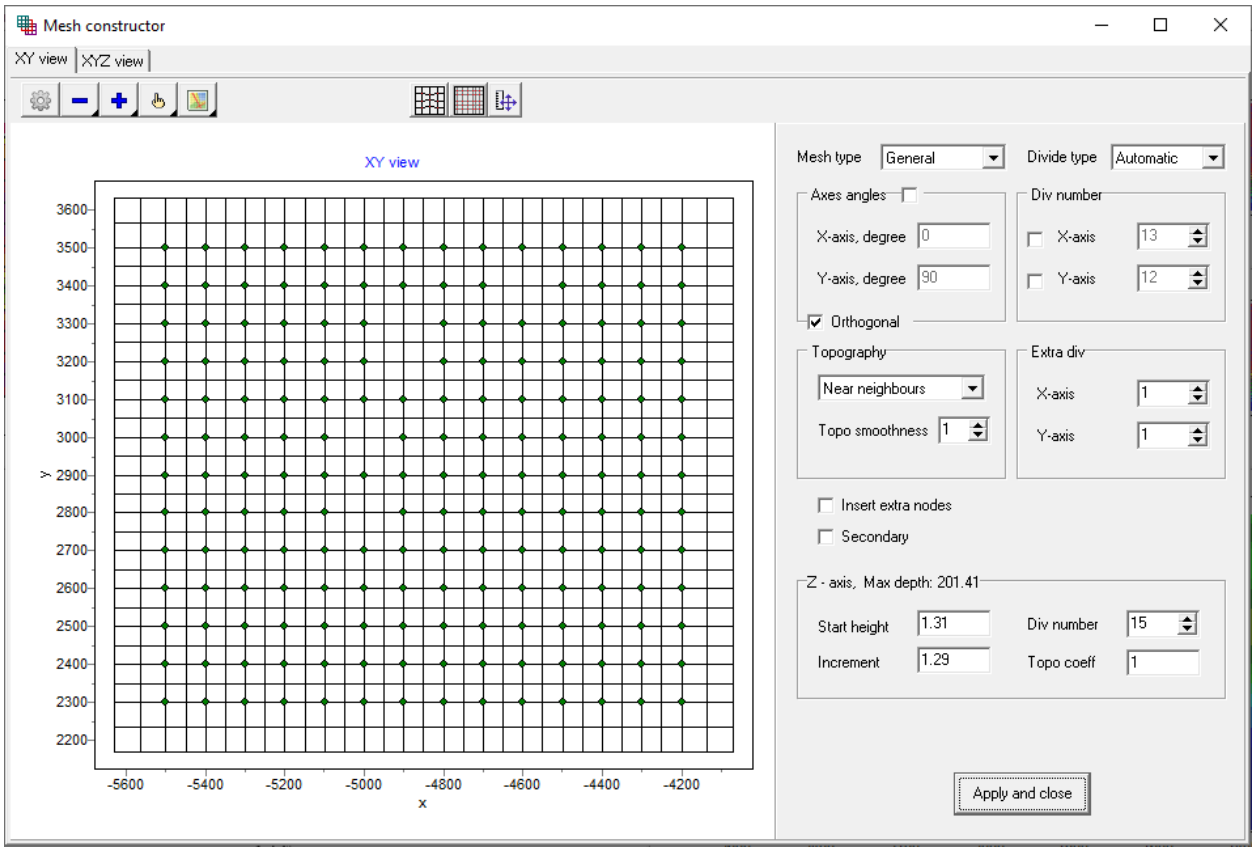
Automatic selection of the vertical mesh

Depths are calculated according to the scheme of the ZondTEM1D program:

- from the times of the first and last gates and the mean apparent resistivity, the range of depths to which the configuration over a homogeneous half-space is sensitive is estimated;
- the thickness of the first cell is limited to 0.2–10 m, the maximum depth to 20–2000 m;
- the increase factor is chosen so that the sum of thicknesses reaches the maximum depth; if the factor exceeds 1.3, the number of layers is increased (up to 100), and if the depth is reached with a margin at a factor of 1, it is decreased (not less than 5).

The found values are written to the **Start height**, **Increment** and **Div number** fields and can be changed manually.

XY view tab



Mesh constructor window, XY view tab

Toolbar

Button	Purpose
	Rebuild the mesh with the new settings. For large surveys this may take time
	Row deletion mode: clicking a node deletes a row of nodes. Right button switches the direction (X / Y)
	Row addition mode: a click adds a row of nodes. Right button switches the direction (X / Y)
	Editing nodes with the mouse (dragging). Right button opens a menu: **Correct mesh** (fix overlapping cells) and **Autocorrection** (fix automatically on every change)
	Left button shows/hides the topography map. Right button opens a menu: **Import topography** (load topography from an XYZ file), **Remove topography**, **Save topography**, **Satellite topography** (load topography from a server by UTM coordinates)
	The **Trapezoidal mesh** window builds a mesh along lines passing through the stations (for curved profiles). Right button deletes the trapezoidal mesh
	The **Simple grid maker** window builds a simple regular mesh defined by lines. Right button deletes such a mesh

Button	Purpose
	Plot scale: 1:1 or fit to window
 and the angle field	Rotate the survey geometry by the given angle (convenient for surveys oriented NW–SE) and undo the rotation

Mesh parameters

Element	Purpose
Survey type	Survey type: **Regular**, a system of parallel profiles; **General**, an irregular survey (the mesh is built according to station density); **One line**, a single profile (to keep the model homogeneous across the profile, enable **Options / Model / 2D mode**)
Automatic / Manual	The number of XY nodes is chosen automatically or set manually
Orthogonal	The main mesh directions are perpendicular
Axes angles (check box and fields)	Main mesh directions (0–180° from the X axis counterclockwise). They are chosen automatically; if necessary, enable the check box and set them manually
Div number (X, Y)	Number of divisions of the base mesh in each direction. Chosen automatically; the adjacent check box allows manual input. In Manual mode a right click on the field opens input of the axis step
Extra div (X, Y)	Number of intermediate nodes between base nodes. A denser mesh is more accurate but requires more time and memory (1 is recommended)
Min / Max	Mesh limits along the directions (Manual mode, check box on). **Alt** + click on the map writes the cursor coordinate to the focused field
Insert extra nodes	Add nodes to equalize the mesh step
Secondary	Mesh for the secondary scheme: nodes are shifted to the cell centres of the base mesh, stations end up inside cells
Topography / smoothing	Number of topography smoothing cycles (0 means no smoothing)
Topography / interpolation	Method of interpolating topography to the mesh nodes: Triangulation, Krigging, Near neighbours

Vertical mesh (Z - axis)

Element	Purpose
Start height	Thickness of the first (top) layer, m
Increment	Ratio of a layer thickness to the thickness of the overlying one
Div number	Number of layers in depth

Element	Purpose
Topo coeff	Attenuation of topography with depth: 0 means all layers follow the surface topography; the larger the value, the flatter the deep layers

The depth of the mesh bottom equals

$$H = h_1 \frac{k^N - 1}{k - 1},$$

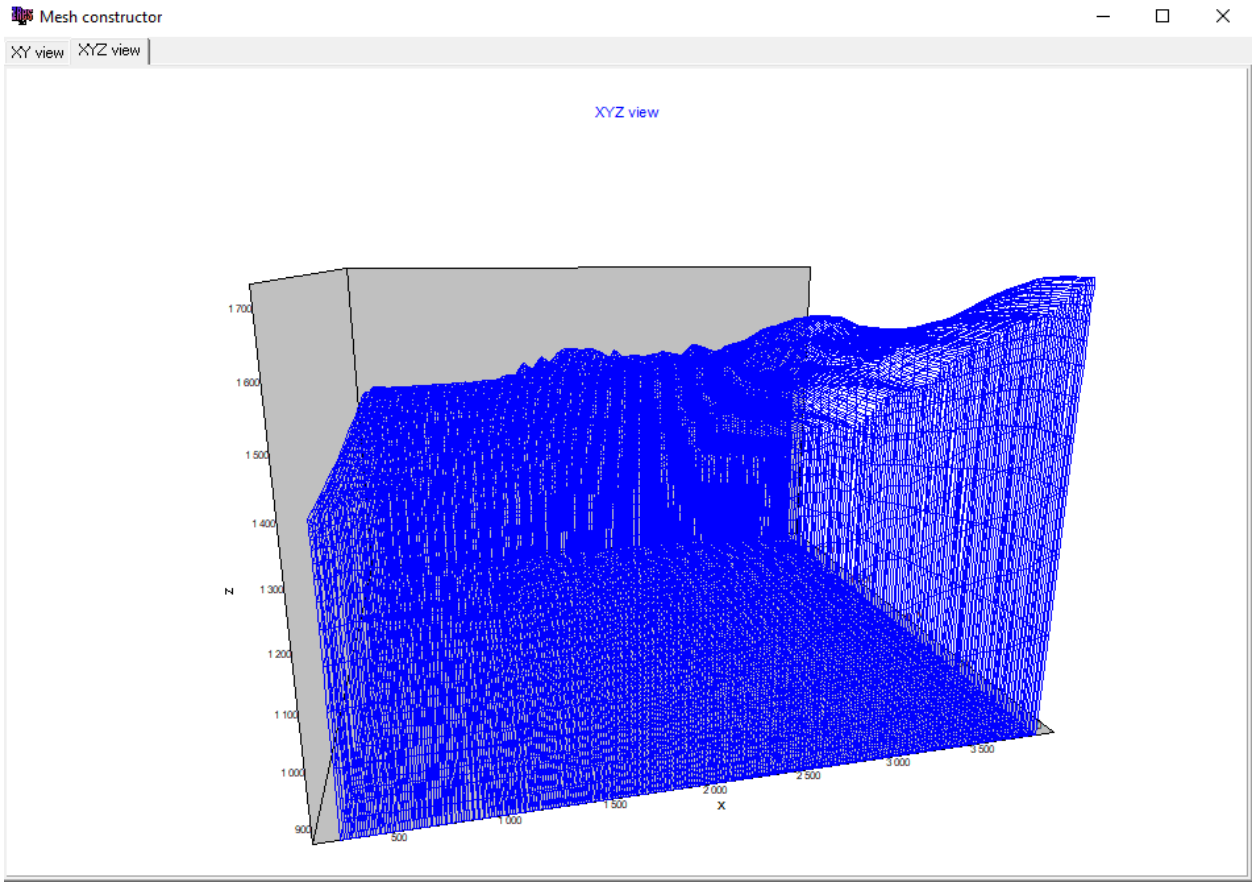
where (h_1) is **Start height**, (k) is **Increment**, (N) is **Div number**.

Mouse in the XY view window

Action	Result
Dragging with the left button down-right / up-left	Zoom into an area / restore the scale (outside editing modes)
Click on a node (deletion/addition modes)	Delete / add a row of nodes
Right button (deletion/addition modes)	Change the row direction X ↔ Y
Dragging a node (editing mode)	Move the node
Alt + click (Manual, Min/Max check box)	Write the coordinate to the active Min/Max field

XYZ view tab

Three-dimensional image of the mesh and topography. Left button rotates, the mouse wheel zooms.



Mesh constructor window, XYZ view tab

Trapezoidal mesh and Simple grid maker windows

Trapezoidal mesh builds a mesh from two families of lines passing through the stations. Each station is assigned line indices i1 and i2; mesh cells are formed by quadrilaterals between neighbouring lines.

Button / element	Purpose
Add line (**Ctrl+A**)	New line of the current direction
Auto line search	Find the optimal line in the selected direction; right button selects Vertical / Horizontal. Repeat if necessary
Delete line / all lines	Delete the current line / all lines
Direction change	Switching between the two families of lines
Line index, labels	Number of the current line; labels at nodes: none, i1, i2 or both
Assignment tool	Left button assigns the current line to a node, right button removes it
Add/delete node	Left / right button
Rotation ±1°	Rotate points (**Ctrl** + click, by 0.1°); undo rotation
Build and check	The result is shown in the status bar
Use mesh	Pass the built mesh to the constructor

Simple grid maker builds a regular mesh by lines: automatic search for the optimal grid (several repetitions may be required), switching directions, line addition mode by click, changing the direction angle by ±1° (**Ctrl**, 0.1°).

Messages when creating the mesh

After pressing **Apply and close** for large meshes with intermediate nodes (Automatic mode) the program asks: “Would you like to group intermediate cells? this reduce Npar in N times”. If the answer is **Yes**, cell grouping is enabled (**Program setup / Options / Cell grouping**): intermediate cells are combined with the base ones into one inversion parameter, which reduces the number of parameters N times without changing the forward problem mesh.

Workflow: building the mesh

1. Check the survey type (**Regular**, **General** or **One line**).
2. Press  and evaluate the mesh: stations should be at nodes (or at cell centres with **Secondary**), without strongly elongated cells.
3. If necessary, correct the directions or the number of divisions, or delete superfluous rows.
4. Load topography (right click on ) and select the interpolation method.
5. Check the vertical mesh: the bottom should be deeper than the maximum depth of investigation, the first layer no thicker than 1/3 of the minimum depth.
6. Press **Apply and close**; accept cell grouping for large models.

Model and its editing

Model cells, parameters and constraints

The ZondTEM3D model is a three-dimensional mesh (see the section on mesh construction): mesh nodes form vertical “columns” under each plan point, and cells (blocks) are located between neighbouring nodes. The number of cells along each axis is one less than the number of nodes. The top row of nodes follows the topography, the underlying rows are offset from it by the depths of the nodal planes (**z_grid**); with depth the influence of topography gradually decays according to the mesh topography coefficient (topoc).

Each cell stores:

Quantity	Meaning
Parameter value **P**	Resistivity, Ohm·m (range 0.001–1 000 000 Ohm·m).
Min, **Max**	The range within which the cell parameter may vary during inversion. The pair 0 and 0 means “no constraints”; in dialogs it is shown as an asterisk ⋆.

Constraints are saved in the project and in the MOD3D file. If a cell value is set outside its range, the range is automatically extended to the new value, so a constrained cell is never “outside” its limits.

The model is edited by slices: the model editor shows one slice (XZ, YZ or XY), and all changes in the editor are immediately transferred to the corresponding cells of the three-dimensional model, both values and constraints. If the **Options / Model / 2D mode** mode is enabled (intended for the case of a single-line survey), after each change the current slice is copied to all slices along the perpendicular axis, i.e. the model becomes two-dimensional (elongated along this axis).

> **Attention.** In **2D mode** the whole current slice, including constraints, is copied to all slices, not only the changed cells. Objects set earlier in other slices are overwritten.

Start model and constraints dialog

Purpose

The dialog is opened with the **Options / Model / Set 1D model** command and serves to quickly set a horizontally layered (one-dimensional) starting model and parameter ranges versus depth. Typical tasks:

- set the starting inversion model from the results of a 1D inversion (for example, in ZondTEM1D), logging or a known section;
- set allowed resistivity limits for each layer, for example, forbid the inversion to obtain a resistivity lower than the known one in the bedrock;
- build a background layered model for modeling, into which local bodies are then inserted in the model editor.

The layered model is applied to the whole mesh, including the edge cells.

What is shown on opening

On opening, the program analyses the current model by horizontal layers of cells and plots on the **Distribution p vs depth** graph (horizontal axis is the parameter, vertical axis is the depth `z, m` from the surface):

- yellow band: the full range (minimum–maximum) of values of the current model at each depth;
- blue band: the mean value of the cell layer ± the mean absolute deviation from it (on a logarithmic scale);
- black step line: the layered model being set;
- two constraint lines: the minimum and maximum parameter limits.

The initial model is a single layer: its value equals the mean (geometric mean) of the mean values of all cell layers. The initial tolerance in the field next to the **Set limites +/-** check box is also computed from the spread of the current model: for resistivity this is the mean relative spread in percent (not less than 20 %). Thus, if the current model is a homogeneous half-space, the tolerance will be minimal, and for an already obtained inversion result it will correspond to the real variability of the model.

Window elements

Element	Purpose
Graph **Distribution p vs depth**	Shows the distribution of the current model parameter versus depth and the layered model being set. The model and constraint lines can be dragged with the mouse (see below).
Layer table	Columns: **#**, layer number; **Z**, depth of the layer top, m; **H**, layer thickness, m; **P**, parameter value; **min**, **max**, parameter limits. The top of the first layer is always 0; the thickness of the last layer (half-space) is not specified.
Number of layers	Number of layers, from 1 to 300; the initial value is 1. When the number is increased by one, a new layer is inserted after the row selected in the table (by halving the layer); its value and limits are copied from the neighbouring layer.
Set limites +/- (check box and field)	If the check box is on (default), new layers are immediately assigned limits relative to the value: the field is given in percent (label **Set limites +/-, %**), the limit is computed as $P/(1 + d/100)$ … $P \cdot (1 + d/100)$. The limits do not go beyond the allowed parameter range.
Apply	Build the model and constraints in the mesh cells and close the dialog.

Editing the table and graph

Values are entered in the table with validation:

- **Z**: the top depth must be greater than the top of the overlying layer and less than the top of the underlying one (with a gap of 0.01 m); thicknesses are recalculated automatically;
- **H**: thickness from 0.01 to 10 000 m; top depths are recalculated;
- **P**: within the allowed parameter range; if the new value goes beyond the layer limits, the limit is shifted to the value;
- **min** not greater than **P**, **max** not less than **P**. Entering a non-numeric value (for example ⋆) in any of these columns removes both limits of the layer.

The **Ctrl+V** combination in the table pastes a column of values from the clipboard (for example, copied from Excel) starting from the current cell; the number of table rows does not change, and values that fail validation revert to the previous ones.

On the graph the lines can be dragged with the left mouse button. If a vertical segment of the black line is grabbed, the layer parameter value changes (within its limits); vertical segments of the min/max lines change the limits (min not greater than the value, max not less). A horizontal segment of the black line moves the boundary between layers in depth (within the neighbouring boundaries). Over a line the cursor changes to a double arrow showing the direction of movement.

What happens when Apply is pressed

The layered model is transferred to the mesh by depth from the surface, i.e. the layers follow the topography. For each horizontal layer of cells three points in depth are taken (at 1/4, 1/2 and 3/4 of the cell thickness):

- the cell value is the mean of the layered model values at these points (geometric mean), so a cell crossed by a boundary receives an intermediate value;
- the lower cell limit is the smallest, the upper the largest of the layer limits at these points; if limits are not set at two or more points, the cell remains unconstrained;
- the cell value is brought within its limits.

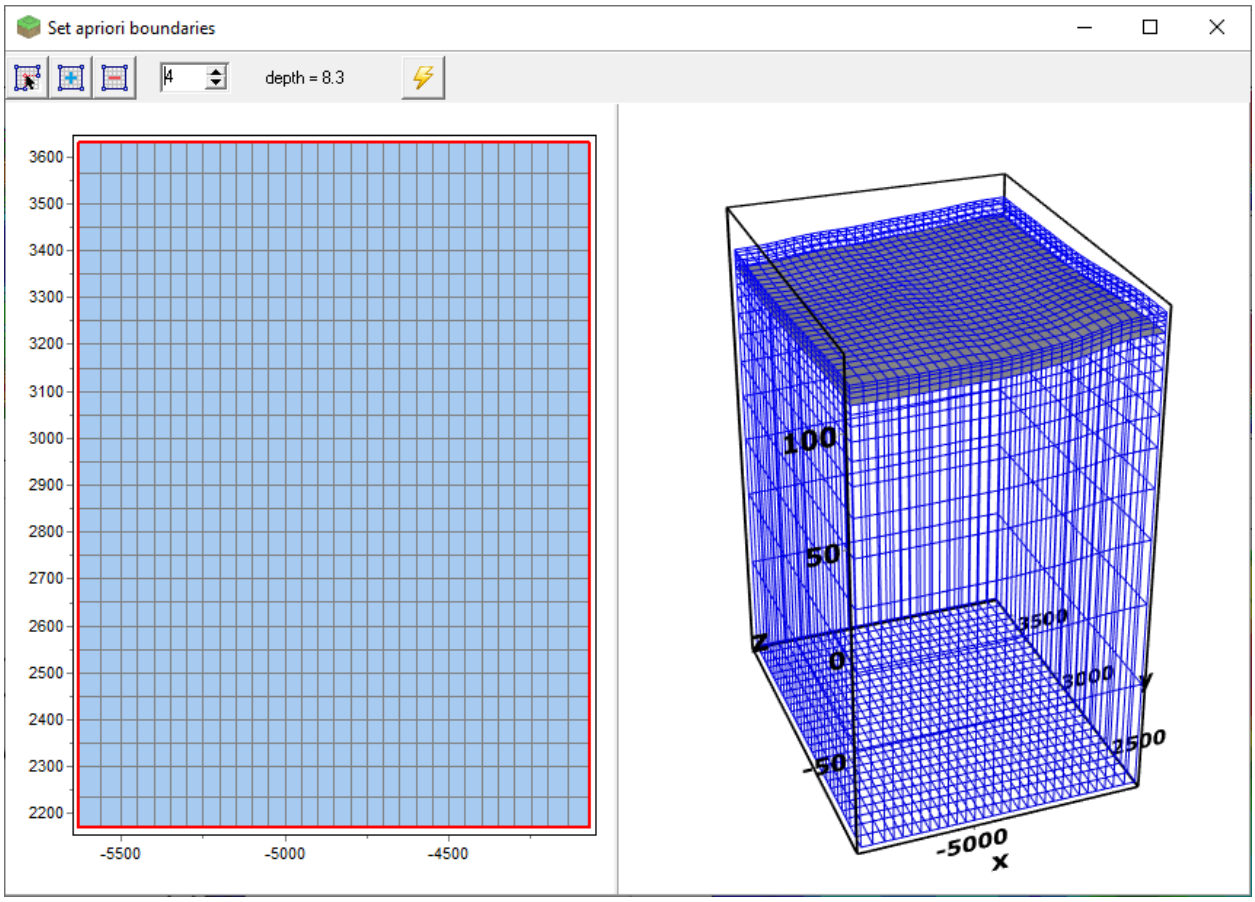
New values and constraints are written to all model cells (existing bodies are erased), and the colour scale range is extended to the specified limits.

> **Tip.** To set constraints only for part of the section (for example, only for the bedrock), leave ⋆ in the min/max columns of the upper layers. To remove all constraints, turn off the **Set limites +/-** check box before adding layers or enter ⋆ in the min/max columns.

Workflow

1. Open **Options / Model / Set 1D model**. Use the bands of the graph to assess how the parameter is distributed in the current model.
2. Set the tolerance in the **Set limites +/-** field (or turn off the check box if constraints are not needed).
3. Increase **Number of layers**, selecting in the table the row below which a new layer is needed.
4. Enter depths (**Z**) or thicknesses (**H**), **P** values and, if necessary, **min**/**max**; refine the line positions with the mouse.
5. Press **Apply**. Check the model on slices and in the 3D window; if necessary, add local bodies in the model editor.

Set apriori boundaries window



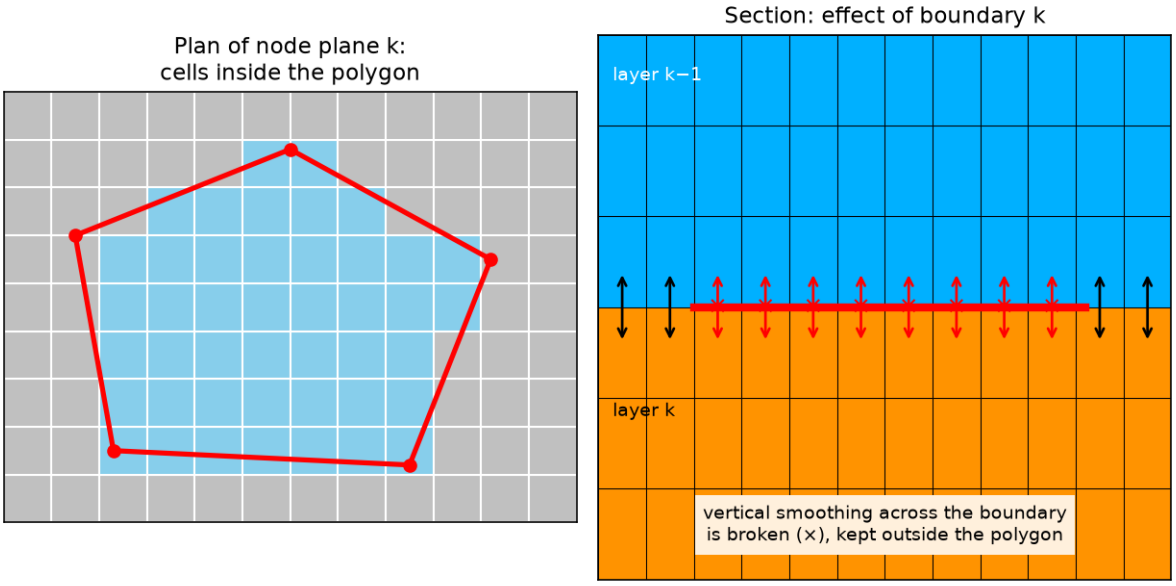
Set apriori boundaries window

Purpose

The window is opened with the **Options / Inversion / Set apriori boundaries** command and sets a priori (structural) boundaries: surfaces on which smoothing between overlying and underlying cells is broken during inversion. Thanks to this, the inversion can build a sharp jump of the parameter at the boundary rather than a smooth transition. Typical applications:

- a bedrock top or groundwater level known from drilling or seismic data that coincides with one of the horizontal mesh planes;
- a boundary that exists not in the whole area but only in part of it (for example, under a sediment cover in some area), specified by a polygon.

The boundary is tied to the nodal plane of the mesh with number k (planes are numbered from the surface; the surface and the mesh bottom cannot be boundaries) and therefore follows the geometry of this plane, including topography. Arbitrary surfaces not coinciding with mesh planes cannot be set: for this the mesh is built so that one of its planes passes at the required depth.



Effect of an a priori boundary: cells of the nodal plane inside the polygon (left) and weakening of vertical smoothing across the boundary (right)

How the boundary is used in the inversion. Boundaries are passed to the Full 3D inversion computational module. Plane k is the top of cell layer k (layers are counted from the surface starting from zero), so the boundary separates layers k−1 and k. When the smoothing operator is built, the weight of the vertical link of cell (i, j, k) with the overlying cell (i, j, k−1) is multiplied by 0.01 if cell (i, j) of plane k is marked as a boundary. The inclined links of the diagonal filter (**Diagonal filter**) passing through plane k are weakened in the same way if both cells of the plane are marked, the current one and the neighbouring one in the link direction. Horizontal links within a layer and links outside the polygon do not change.

- The boundary acts only in algorithms with smoothing (**Occam**, **Smoothness constrained**, **Focused**); in **Marquardt** and **Blocks** there is no smoothing operator, and the boundary has no effect.
- In **Pseudo 3D** inversion boundaries are not used.
- Boundaries are kept until the mesh is rebuilt, another project is opened or the dialog is cancelled; they are not written to the project file. Before repeating the inversion after opening a project, the boundaries must be set again.

Window elements

Button / element	Purpose
Add user defined polygon	Enable the polygon drawing mode on the plan: each left click adds a vertex, a right click (after at least two vertices) adds the last vertex and closes the polygon. The polygon replaces the previous boundary of the current plane.
Add whole plane	Create a boundary over the whole plane (polygon along the mesh outline).
Remove current boundary	Delete the boundary of the current plane.
Boundary number field (counter on the toolbar)	Number of the nodal plane for which the boundary is set (from 1 to the number of planes − 2). When the

Button / element	Purpose
	number changes, the boundary of the selected plane is shown; the label on the right is the mean depth of the plane (`depth = …`).
Copy boundaries to project and close	Pass the boundaries to the project and close the window.
Plan (left part)	Mesh in plan for the selected plane. Cells included in the boundary are filled light blue, the others grey. Outside the drawing mode the plan can be zoomed and panned with the mouse.
3D view (right part)	Mesh and boundaries in 3D; the mouse wheel zooms.

A cell is considered to belong to the boundary if more than half (at least 5 of 9) of the cell control points are inside the polygon. Therefore it is sufficient to draw the polygon with cell accuracy.

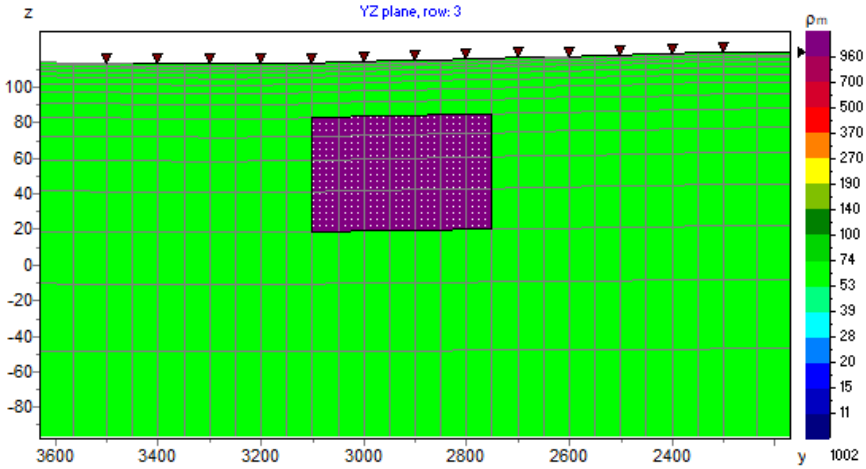
> **Attention.** Closing the window with the cross (without the **Copy boundaries to project and close** button) deletes all a priori boundaries, including those set earlier. Boundaries are also reset when the mesh is rebuilt and when a project is opened.
>
> **Tip.** A priori boundaries work best with smooth (L2) inversion, where it is smoothing that blurs contrasts. The boundary must be justified: an incorrectly set boundary creates a false jump in the model.

Workflow

1. Build the mesh so that one of the horizontal planes passes at the depth of the known boundary.
2. Open **Options / Inversion / Set apriori boundaries**.
3. Select the plane number in the boundary number field on the toolbar, using the `depth = …` label.
4. Press **Add whole plane** for a continuous boundary, or **Add user defined polygon** and outline the area (a right click closes the polygon).
5. Repeat for other planes if necessary.
6. Press **Copy boundaries to project and close**.
7. Select an algorithm with smoothing (**Occam**, **Smoothness constrained** or **Focused**) and run **Full 3D inversion** (right click on the inversion button).
8. Check the result on slices across the boundary: the contrast at plane k should become sharper, while outside the polygon the model remains smooth.

Model editing

Model editor



Model editor: YZ slice

The geoelectric model is edited in the ***model editor***, the lower right graphic section of the main window in **Options / Model / Block-section** mode. The editor is used to change the parameters of individual cells and groups of cells with the mouse and keyboard. To the right of the model area there is a colour scale linking colour to parameter value; below the scale the current "brush" value is shown, i.e. the value that will be assigned to cells.

The slice orientation is selected in the **Options / Model / Plane** menu (XZ, YZ, XY) or with hot keys:

- **Alt+1**: slice in the XZ plane;
- **Alt+2**: slice in the YZ plane;
- **Alt+3**: slice in the XY plane.

The mouse wheel over the editor switches to the neighbouring slice (forward or backward along the perpendicular axis), so a body can quickly be "drawn" in several neighbouring slices.

Working with cells is similar to editing a raster image in graphics editors. As the cursor moves over the model, the status bar of the main window shows the cursor coordinates, the cell number (**cellX**, **cellY**), its value (**Value**) and, if set, its limits (**Min**, **Max**), as well as the current editor mode (**Editing**, **Rect selection**, etc.). The active cell is outlined by a rectangle, the cursor. Selected cells are marked with a hatch of the selection colour.

Editor modes and cell context menu

A right click on the model area opens the cell menu:

Item	Purpose
Display cell setup	Open the **Cell setup** dialog for the active cell or selection.
Cell to cursor value	Make the value of the active cell the current brush value (same as the **Z** key).
Edit mode	Drawing mode: a click or drag with the left button assigns the current value to cells.
Selection / Free-form selection	Free-form selection: dragging with the left button

Item	Purpose
	outlines a contour.
Selection / Rectangular selection	Rectangle selection (by dragging from corner to corner).
Selection / Elleptical selection	Ellipse selection inscribed in the dragged rectangle.
Selection / MagicWand	“Magic wand”: a click selects the active cell and the connected neighbouring cells with similar values. The degree of similarity is set by the **Selection admissibility, %** parameter in the **Model setup** dialog.
Selection / Remove selection	Clear the selection (same as the **V** key).
Clear model	Clear the current slice: all cells of the slice are assigned the value of the host half-space, constraints are removed.

When drawing (**Edit mode**) the actions depend on modifier keys:

- left button: assign the current value to the cell;
- **Shift+left button**: increase the cell value by 10 %;
- **Shift+right button**: decrease the cell value by 10 %.

If the click is made on a selected cell, the operation is applied to the whole selection at once: this makes it convenient to fill the selected area with one value or to proportionally strengthen/weaken the contrast of a body.

In selection modes, holding **Shift** while dragging adds a new area to the already selected cells (without Shift the previous selection is cleared).

The selection is moved and copied with the **Ctrl** key held down: the cursor is placed on a selected cell and the selection is dragged with the mouse. **Ctrl+left button** moves: the contents of the selected cells are cut and pasted to the new place, and the vacated cells receive the host value. **Ctrl+right button** copies: the original cells are kept. Moving is performed within the current slice.

If the right button is pressed together with **Alt**, the context menu does not open and cell values do not change.

The current brush value is also set by a left click on the colour scale (the editor then switches to **Edit mode**) or with the ↑/↓ keys when the cursor is over the scale (multiplication/division by 1.1).

Changes made in the editor are immediately transferred to the three-dimensional model and taken into account in the 3D window.

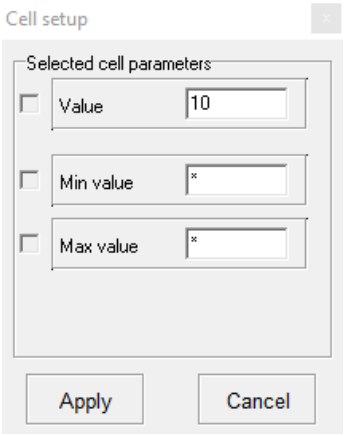
Editor keys

Action	Combination	Where it works
XZ / YZ / XY slice	**Alt+1** / **Alt+2** / **Alt+3**	main window
Neighbouring slice	mouse wheel	over the model editor
Moving the cursor over cells	← → ↑ ↓	cursor over the model
Brush value ×1.1 / ÷1.1	↑ / ↓	cursor over the colour scale
Assign the brush value to the active cell (or selection)	**Insert**	model editor

Action	Combination	Where it works
Assign the host value to the active cell (or selection)	**Delete**	model editor
Take the value of the active cell into the brush	**Z**	model editor
“Magic wand” from the active cell, adding to the selection	**X**	model editor
Clear selection	**V**	model editor
Select / deselect the active cell	**S**	model editor
Assign the brush value	left button	**Edit mode**
Increase / decrease the value by 10 %	**Shift+left** / **Shift+right** button	**Edit mode**
Add an area to the selection	**Shift** + drag	selection modes
Move / copy the selection	**Ctrl+left** / **Ctrl+right** button + drag	on a selected cell
Set the brush value	left button	colour scale
Context menu	right button	model, scale, margins around the model
Right button without a menu and without changing the model	**Alt+right** button	model

Cell setup dialog

The dialog is opened with the **Display cell setup** item and allows exact values of the parameter and its limits to be entered for the active cell or for the whole selection.



Cell setup window

Element	Purpose
Value	Parameter value.
Min value, **Max value**	Limits of parameter variation in the inversion. * means limits are not set.
Check boxes to the right of the fields	Available if the active cell is selected: they mark which of the quantities (value, lower, upper limit) to apply to all selected cells. All are checked by default.
Apply	Apply and close the dialog.

Element	Purpose
Cancel	Close without changes.

Input features:

- if one of the **Min value** / **Max value** fields contains ⋆ (or a non-numeric value), the cell constraints are removed;
- if **Min value** is greater than **Max value**, the limits do not change;
- a value beyond the limits automatically extends them;
- the value is brought to the allowed parameter range.

Constraints are especially useful in inversion: for example, a narrow resistivity range can be set for a unit known from drilling, and a wide one for a presumed body.

Model and colour scale menu

A right click on the margins around the model opens the display menu:

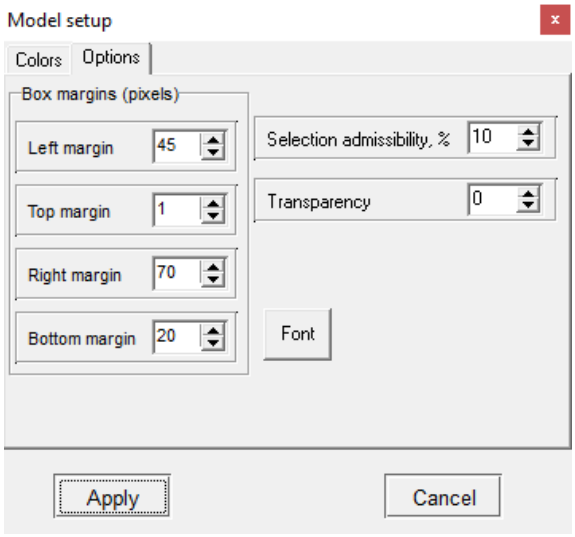
Item	Purpose
Display model mesh	Show mesh lines.
Display objects border	Show boundaries between cells with different values (body outlines).
Setup	Open the **Model setup** dialog.
Display color bar	Show the colour scale.
Zoom&&Scroll	Zoom and scroll mode: dragging with the left button down-right zooms into the selected area, up-left restores the original scale. While the mode is on, editing cells with the mouse is disabled.
Print preview	Preview and print the model.
X:Y 1:1	Equal scale on the horizontal and vertical axes.

Right click on the colour scale:

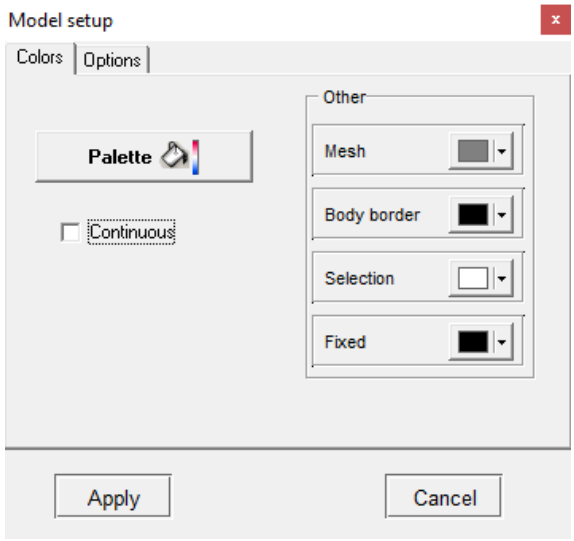
Item	Purpose
Set range	Set the minimum and maximum scale values.
Automatic	Determine the scale limits from the minimum and maximum values of the model parameter.
Log scale	Logarithmic scale.
Smooth image	Show the model with a smooth interpolated fill. In this mode cell editing is disabled.
Set cursor value	Enter the brush value from the keyboard (within the scale).

Model setup dialog

The **Setup** item of the model menu opens the colour and view settings of the editor.



Model setup window, Options tab



Model setup window, Colors tab

Element	Purpose
Colors tab	Colours of image elements.
Palette	Open the palette settings of the colour scale. The palette window plots a histogram of the value distribution of the current model slice over 50 scale intervals; this helps to choose the limits and the distribution of colours.
Mesh	Colour of mesh lines.
Body border	Colour of boundaries between contrasting cells (body outlines).
Selection	Colour of the selection hatch; the **Inverted** option displays the selection by inverting the cell colours.
Options tab	Image parameters.
Left margin, **Top margin**, **Right margin**, **Bottom margin**	Image margins in pixels, 0–100, 0 by default. Increase them if axis or scale labels do not fit in the window; the left and right margins are also applied to the pseudosections of the main window so that their axes

Element	Purpose
	coincide with the model.
Selection admissibility, %	“Magic wand” tolerance, 0–200 %: the larger it is, the more the values of the cells included in the selection may differ.
Transparency	Transparency of the model image, 0–80 %. Useful if a background (geological section, image) is loaded under the model.
Font	Font of the colour scale labels.
Apply	Apply the settings: update the palette and the model image. The window remains open.
Cancel	Close the dialog.
Default	Restore default settings (applied after **Apply**).

Workflow

1. Set the background: homogeneous or layered (**Options / Model / Set 1D model**).
2. Select the slice plane (**Alt+1/2/3**) and the required slice with the mouse wheel.
3. Select the brush value by clicking the scale or with the **Z** key on a cell with the required value.
4. Select an area (rectangle, ellipse, free form, magic wand) and fill it by a left click on the selection or with the **Insert** key.
5. Repeat in neighbouring slices (mouse wheel) so that the body gets volume. For elongated bodies use **2D mode**.
6. If necessary, set the values and limits exactly in **Cell setup**.
7. Check the model in the 3D window and run the forward problem.

Forward problem and inversion

Forward problem

The  button (**Calculate forward problem**) computes the response of the current 3D model for all stations. The calculation is always performed by the full three-dimensional finite element method in the external 64-bit computational module (zondtem3d.dll), regardless of the selected inversion type.

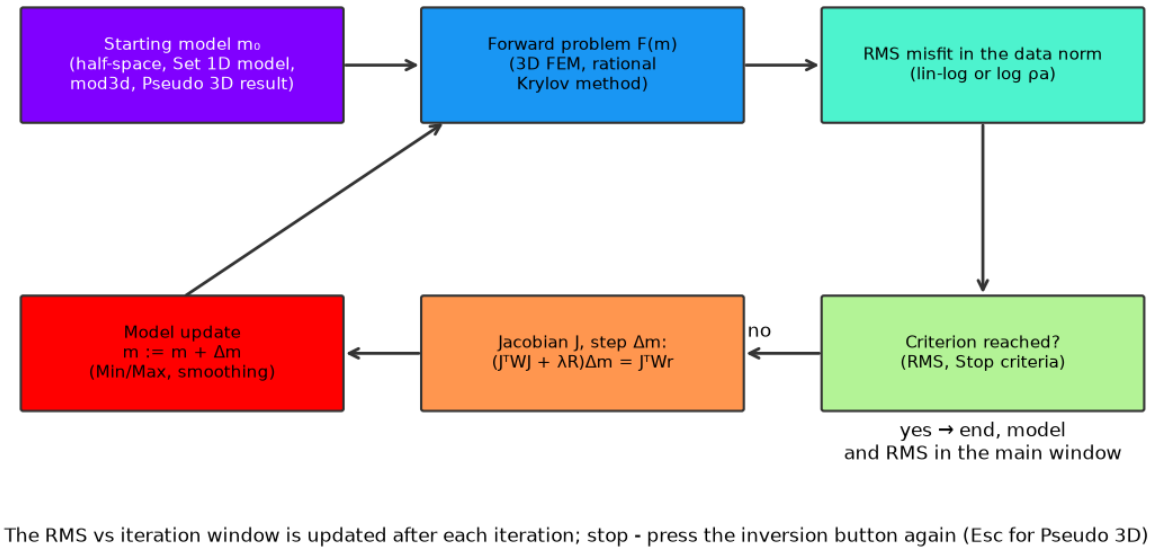
Calculation scheme:

1. The program writes the model, mesh, loop geometry, times and pulse shape to a task file (calc64.bi1 in the program folder) and starts the computational module.
2. The module builds a finite element mesh (model mesh plus outer cells) and computes the field of each transmitter loop. The time dependence is computed by the rational Krylov method: the matrix is factorized for a small set of complex shifts, after which the response at all times is obtained without step-by-step time integration.
3. The result (file resp64.bi1) is read by the main program; the calculated data appear on the pseudosections and in the **Sounding curves** window, and RMS is shown in the status bar.

Calculation progress is shown by the indicator in the status bar; the stages (mesh building, factorization, solution) are shown in the message section. The calculation time depends mainly on the number of mesh nodes and the number of transmitter loops.

> **Tip.** If the calculation is interrupted every time without a message, run **Help / Restart library** and check whether an antivirus blocks the computational module.

Inversion types



Iterative inversion scheme

The inversion type is selected by a right click on the inversion button ; the left button starts the selected type. During the calculation the button changes to  ; pressing it again stops the inversion.

After stopping or completion the program reports the calculation duration in the status bar ("Inversion took …").

	Pseudo 3D inversion	**Full 3D inversion**
Forward problem	1D (layered earth) beneath each station	3D FEM
Speed	Seconds to minutes	Minutes to hours
Accounting for 3D effects	No (only lateral smoothing)	Yes
Where it runs	In the main program	In the external 64-bit module
Stopping	Inversion button or **Esc**	Inversion button
Purpose	Quick estimate, starting model for Full 3D	Final model

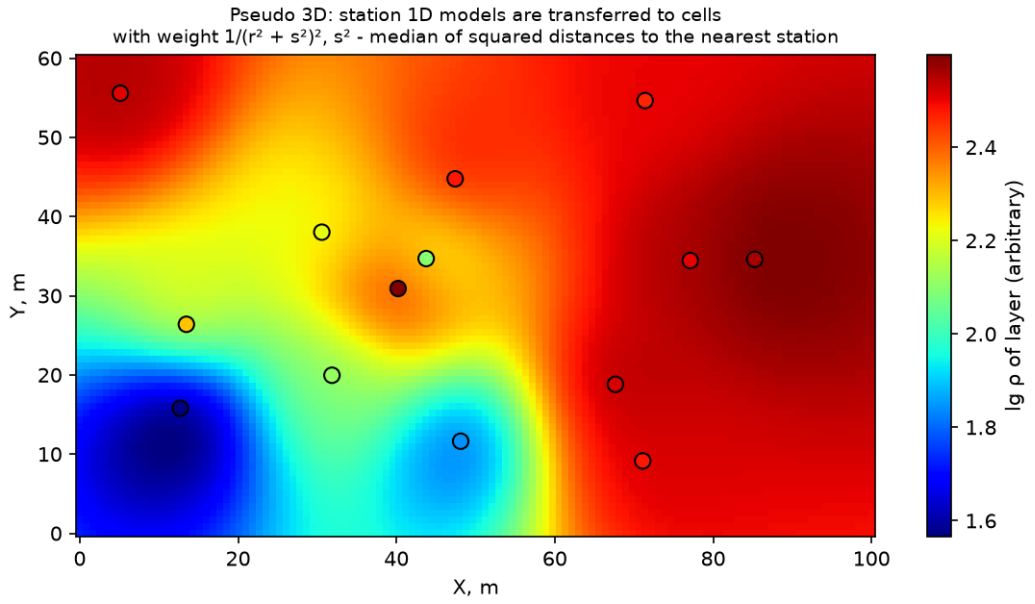
Pseudo 3D

A layered model is built beneath each station; layer thicknesses equal the mesh-average thicknesses of the horizontal cell layers, the last layer is a half-space. All 1D models are fitted jointly: regularization couples neighbouring layers of one station and identical layers of neighbouring stations. The forward problem is the same as in the ZondTEM1D program (taking into account the pulse shape, finite loop and turn-off time).

After each iteration the 1D models are transferred to the cells of the 3D mesh with the weight

$$w = \frac{1}{(r^2 + s^2)^2},$$

where (r) is the distance from the cell centre to the station, (s^2) is the median of the squared distances between stations and their nearest neighbours. This smoothed interpolation eliminates "pits" around individual stations.



Transfer of station 1D models to the 3D mesh

Pseudo 3D uses the main inversion settings: algorithm, smoothing reference model (Median / Start model / Previous), smoothing norm, Robust, Stabilizer, sensitivity normalization, Line search, focusing parameters and data norm. Stations excluded with the **Invert** check box in the **Sounding**

curves window are not included in the misfit, but their models are formed by smoothing with neighbours.

Full 3D

Each iteration includes the 3D forward problem and the calculation of sensitivities (Jacobian) in the external module. All inversion settings are passed to the module: algorithm, reference model, norms, Robust, Stabilizer, sensitivity normalization, Lim based inv, Diagonal filter, automatic selection of the smoothing parameter, number of iterations, limits, Smoothing factor, focusing parameters, Smoothness ratio, Line search and Dec factor, cell grouping and the number of surface layers, as well as the layered term (**Layered term**) and a priori boundaries (**Set apriori boundaries**). In Pseudo 3D the layered term and a priori boundaries are not used.

> **Tip.** Run Pseudo 3D first: the result will become the starting model for Full 3D and will reduce the number of expensive 3D iterations. Save the intermediate result to the buffer (**Buffer / Model 1**) to compare models later.

Program setup window

The **Program setup** dialog contains the inverse problem parameters (**Inversion** and **Options** tabs). It is opened with a toolbar button or the **Options / Program setup** menu item. The dialog is modeless: while the window is open one can work with the model, and when cell grouping is being set up the inversion mesh is drawn immediately over the model slice.

The **Apply** button checks and saves the entered values, reloads them into the form and closes the dialog. Values outside the allowed range or non-numeric values are not accepted; the previous value is kept. The **Cancel** button (or closing the window with the cross) closes the dialog without saving the entered values; changes in the cell grouping table and check boxes affecting the mesh display are applied immediately.

Default values for a new project are given in the tables below.

Inversion tab

Element	Purpose	Default / range
Inversion	Inverse problem solution algorithm: Smoothness constrained, Occam, Marquardt, Blocks, Focused	Occam
Smoothing factor (check box)	Automatic selection of the initial value of the smoothing parameter when the inversion starts. The found value is written to the **Smoothing factor** field, after which the check box is cleared	off
Smoothing factor (list)	Smoothing (damping) parameter μ of the first iteration. List: 0.005, 0.01, 0.02, 0.05, 0.1, 0.2; any value can be entered	0.05; 0.0001–500
Focusing parameters / Threshold	Contrast threshold e for the Focused algorithm (and for the **Options / Inversion / Robust** item)	0.01; 1e-10–5000

Element	Purpose	Default / range
Focusing parameters / Sharpness	Body compactness parameter for Focused	0.1; 0–1
Stop criteria / Iterations (check box and field)	Stop the inversion after the given number of iterations. The field is available when the check box is on	on, 10; 1–150
Robust weighting scheme	Robust data reweighting: measurements with anomalously large misfit receive less weight	on

The **Focusing parameters** group is available only when the Focused algorithm is selected or when the **Options / Inversion / Robust** item is enabled.

Inversion algorithms

Occam: least-squares inversion with a smoothing operator and minimization of the roughness of the model itself [Constable, 1987]. The result is the smoothest parameter distribution compatible with the data.

$$(A^T W^T W A + \mu C^T C)\Delta m = A^T W^T \Delta f - \mu C^T C m$$

Here C is the operator of first differences between neighbouring cells (horizontally and vertically with **Smoothness ratio** weights). The term on the right-hand side penalizes the model itself rather than the update, so the model cannot accumulate false inhomogeneities from iteration to iteration. **Occam is the default algorithm and is recommended for the initial inversion of most data**: it is stable, depends weakly on the starting model and gives a minimum of artefacts. The smoothness of the result grows with increasing **Smoothing factor**; too large values lead to an increase in misfit.

Smoothness constrained: the least-squares method with a smoothing operator applied to the update:

$$(A^T W^T W A + \mu C^T C)\Delta m = A^T W^T \Delta f$$

At each iteration the increment is smoothed rather than the whole model, so the algorithm does not minimize the model contrast and allows lower misfit values to be reached than Occam. The result is a smooth but more contrasting distribution. The algorithm is useful when Occam gives a model that is too blurred and the data are of good quality.

Marquardt: the classical least-squares method with a damping parameter (Ridge regression) [Marquardt, 1963]:

$$(A^T W^T W A + \mu I)\Delta m = A^T W^T \Delta f$$

Only the update magnitude is constrained; there is no coupling between neighbouring cells. With a small number of parameters the algorithm gives a contrasting model, but with a large number of cells the result may be unstable and the misfit may grow (divergence). Marquardt is best applied as a refining step (a few iterations to reduce the misfit) after Occam or Smoothness constrained.

Blocks: fitting the parameters of individual model regions. All cells with the same parameter value are combined into one block, and one value per block is fitted (the system of equations is the same as for Marquardt). The number of parameters is reduced dramatically, so the algorithm is well suited for refining the resistivities of known bodies. Blocks are prepared in advance: either manually in the model editor (regions are assigned different values) or by rasterizing/smoothing the

model after inversion (Focused is best). When the **Program setup** dialog is open, block boundaries are highlighted on the model slice.

Focused: the least-squares method with a smoothing operator and contrast focusing [Portniaguine, 2000]. The result is a piecewise-smooth model: regions with nearly constant resistivity and sharp boundaries between them.

$$(A^T W^T W A + \mu C^T R C)\Delta m = A^T W^T \Delta f - \mu C^T R C m$$

The matrix R contains smoothing weights for each pair of neighbouring cells. The weight depends on the contrast r_i between the cells and the **Threshold** e:

$$R_i = \frac{e^2}{e^2 + r_i^2}.$$

If the contrast is much smaller than the threshold, the weight is close to 1 and the cells are smoothed as in Occam; if the contrast is much larger than the threshold, the weight tends to 0: a boundary is assumed to pass between the cells, and smoothing across it does not act. When Focused is selected, the regularization reference model automatically switches to the median (**Options / Inversion / Smoothness / Median**).

Algorithm	Regularization	Result	When to use
Occam	$\mu C^T C$ on the model	The smoothest stable model	Initial inversion, noisy data, first look
Smoothness constrained	$\mu C^T C$ on the update	Smooth, more contrasting, lower misfit	Good-quality data, refinement after Occam
Marquardt	μI	Contrasting, may be unstable	Short misfit refinement, small number of parameters
Blocks	μI, parameter = block	Values in specified regions	Testing hypotheses, refining known bodies
Focused	$\mu C^T R C$ with a threshold	Piecewise smooth, sharp boundaries	Contrasting bodies (voids, ore bodies, bedrock)

> **Tip.** Start with Occam and the default parameters. If the model is too blurred and the misfit is noticeably above the noise level, decrease Smoothing factor or switch to Smoothness constrained. If contrasting bodies with sharp boundaries are expected, use Focused as the next step, comparing the result with Occam.

Smoothing factor

Smoothing factor is the value of the smoothing (damping) parameter μ at the first iteration. It sets the balance between minimizing the data misfit and the model "misfit" (roughness or update magnitude) and is used in all algorithms: in Occam, Smoothness constrained and Focused as the smoothing weight, in Marquardt and Blocks as the damping parameter. If the **Line search** optimization (**Options / Inversion / Optimization**) is off, the value decreases at each subsequent iteration: **Dec factor** sets how many times the smoothing parameter decreases. With **Line search** optimization on, the program searches at each iteration for the value minimizing the misfit (for Occam, taking into account the model roughness term); this mode often speeds up convergence but requires several times more time and may lead to "getting stuck" in a local minimum.

Selection recommendations:

- 0.05: the optimal initial value for most data;
- 0.01–0.02: for good-quality data when a lower misfit is required;
- 0.1–0.5 and more: for noisy data or if a smoother and more stable model is needed;
- 0.005 and less: only for very good-quality data; instability and false details are possible.

The check box next to the list enables automatic selection of the initial value before the first iteration. The selection does not guarantee the best choice but ensures stable convergence; it usually gives a somewhat overestimated value (within 0.001–0.5). After the calculation the found value remains in the field and the check box is cleared, so a repeated inversion starts from it.

Robust weighting scheme

When the option is on, measurement weights are recalculated at each iteration depending on their misfit: measurements falling sharply out of the general trend receive less weight and affect the model less. This scheme effectively suppresses individual outliers associated with systematic errors (noise, error in loop geometry, wrong receiver area). It is recommended to keep the option on almost always. If there are as many bad data as good measurements, the scheme will not help; the data must be rejected first (quality control window). For very good-quality data the option can be turned off.

Stop criteria

Iterations: when the check box is on, the inversion stops after the given number of iterations (10 by default, maximum 150). In addition, the calculation can be stopped by pressing the inversion button again (for Pseudo 3D, also with the **Esc** key). For large 3D models it is reasonable to run 3–5 iterations first, evaluate the result and continue if necessary: a repeated run starts from the current model.

Focusing parameters

Threshold: the contrast threshold of neighbouring cells e, starting from which their parameters are not averaged (a boundary is assumed between them). The value is chosen empirically:

- 0.1–1: for models with high-contrast boundaries;
- 0.01–0.1: for moderate contrasts;
- very small values destroy the regularization operator and may lead to divergence or the formation of many small “blocks”; in this case the value should be increased;
- very large values eliminate the focusing effect, and the result approaches Occam.

For the Focused algorithm the threshold should usually be about 10 times greater than the value suitable for the **Robust** item of the main menu.

Sharpness: the second parameter of the focusing inversion controlling the compactness (volume) of anomalous bodies, within 0–1. At large values (0.3–1) bodies tend to be smaller in volume and more compact; at small values the distribution remains closer to piecewise smooth. The value is chosen empirically.

Options tab

Element	Purpose	Default / range
Model / Min resistivity	Lower resistivity limit in the inversion: smaller values are clipped. List: 0.01, 0.1, 1, 10	0.1 Ohm·m

Element	Purpose	Default / range
Model / Max resistivity	Upper resistivity limit. List: 100, 1000, 10000	10000 Ohm·m
Model / Smoothness ratio	Ratio of horizontal to vertical smoothing. The icon next to the list shows the shape of the "smoothing ellipse"	0.5; 0.1–10
Model / Number of surface layers	Number of top cell layers (0–3) in which smoothing is weakened	0
Cell grouping (check box)	Enable cell grouping and show the inversion mesh table	off
Table **Layer / X / Y / Z**	Cell grouping parameters by layers	1 / 1 / 1 / layer number

Min / Max. The limits restrict the range in which the program may change the parameter. They protect against non-physical values (for example, resistivities of millions of Ohm·m in zones of low sensitivity). In ordinary inversion values beyond the limits are simply clipped, and the inversion may constantly "hit" the limit, which slows down convergence. If the range is known accurately enough, it is recommended to enable **Options / Inversion / Optimization / Lim based inv**, an inversion variant that takes the limits into account when computing the update. Individual limits for individual cells are set in the model editor (a priori information).

Smoothness ratio. Values less than 1 strengthen horizontal smoothing compared with vertical and are suitable for horizontally layered media; values greater than 1, for media with strong lateral variability (vertical contacts, dykes). In the absence of geological information use 1. Typical values are 0.2–1.

> **Note.** In the current version Pseudo 3D inversion uses its own fixed ratio of 0.5 and does not take the **Smoothness ratio** value into account; for Full 3D the value is passed to the computational module.

Number of surface layers. If the near-surface part of the section is very heterogeneous and contrasting (fill, concrete, utilities), the smoothing operator "drags" these inhomogeneities into the underlying part of the model. The value n sets the number of top cell layers in which smoothing is weakened: the parameters of these cells are more weakly coupled with their neighbours, and near-surface inhomogeneities affect the rest of the model less.

Cell grouping

Grouping applies to Full 3D inversion. The number of 3D inversion parameters equals the number of model cells and for large areal surveys may amount to hundreds of thousands. Memory and the time for computing the Jacobian and solving the system grow with the number of parameters. Cell grouping combines neighbouring cells into one inversion parameter; the forward problem mesh does not change, so the accuracy of the field calculation is preserved. Combining deep cells is natural: the resolution of the method decreases with depth, and the inversion cannot reliably resolve small deep cells anyway. Ideally the number of fitted parameters should be comparable to the number of data.

The table contains a row for each model layer:

Column	Meaning

Column	Meaning
Layer	Model layer number (top down), not editable
X	Number of cells along the X axis combined into one inversion cell (1–19)
Y	Number of cells along the Y axis combined into one inversion cell (1–19)
Z	Layer number of the inversion mesh. Layers with the same Z number are combined vertically into one parameter. For the first layer it is always 1; for each next one it is either the same number as the previous one (combined with it) or 1 greater

Rules the program follows automatically:

- when layers are combined in Z, the X and Y values for all layers of the group are set equal, according to the changed row;
- a change of Z in a row carries the numbering over to all underlying layers;
- the outermost rows of model cells (the first and last columns in X and Y) are not combined into groups; they remain separate parameters, and the remaining cells of the layer are divided into groups of the given width, the last group may be narrower;
- a right click on a cell of the X or Y column opens a value input window; the value is applied to this layer and to all underlying ones;
- after each change the inversion mesh is redrawn over the model slice, and the status bar of the main window shows the number of data records, the number of parameters and the estimate of required memory.

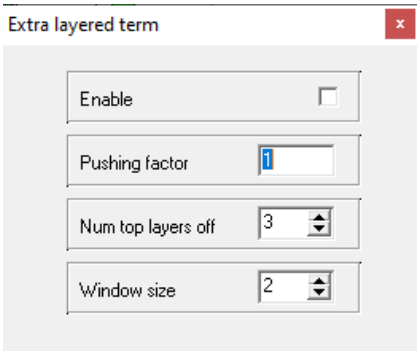
> **Tip.** A typical scheme for a large areal survey: the top 3–5 layers without grouping, deeper 2 cells in X and Y, in the lower third of the model 3–4 cells and combining pairs of layers in Z. Do not combine the top layers in Z: the first layer must be thin enough to approximate the near-surface part of the section.

When a mesh with intermediate nodes is built, the program offers to combine intermediate cells right away (“Would you like to group intermediate cells?”): if accepted, the **Cell grouping** check box is turned on and the corresponding X and Y are set in the table.

Extra layered term dialog

Purpose

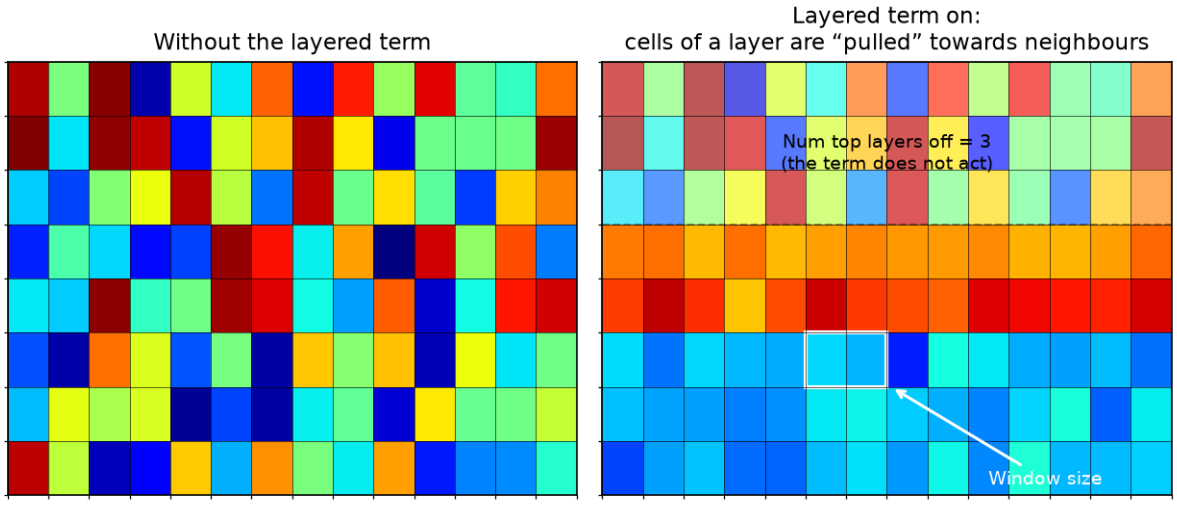
The dialog is opened with the **Options / Inversion / Optimization / Layered term** command and adds an extra “layered” term to the objective function of the Full 3D inversion. It pushes the model towards a horizontally layered structure: the vertical resistivity profiles of neighbouring cell columns become similar to each other. The option is useful if the geology is known to be layered and the ordinary inversion gives a “patchy” model with false local anomalies. If local bodies are expected in the section, the weight of the layered term should be kept small.



Extra layered term window

Element	Purpose	Default
Enable	Enable the layered term. The state is shown by a check mark at the **Layered term** menu item.	off
Pushing factor	Weight of the layered term w; values from 0 to 1000 are accepted. The larger it is, the more strongly the model is levelled horizontally.	1
Num top layers off	Number of top cell layers on which the layered term does not act (0–5). The near-surface part is usually the most heterogeneous, so it is excluded.	3
Window size	Window N (in cells): each column is compared with columns shifted by 0…N cells in X and 0…N cells in Y (except itself).	2

The settings are saved in the project and passed to the computational module at each start of Full 3D inversion. In Pseudo 3D the layered term is not used.



Effect of the layered term (schematic): without it (left) and with it (right)

How the layered term works

For each pair of cell columns (i, j) and $(i + d_x, j + d_y)$, $d_x, d_y = 0 \dots N$, $(d_x, d_y) \neq (0,0)$, over layers k from **Num top layers off** to the mesh bottom the following is computed

$$g = 1 - \frac{\sum_k (m_{1k} - \bar{m})(m_{2k} - \bar{m})}{\sqrt{\sum_k (m_{1k} - \bar{m})^2 \sum_k (m_{2k} - \bar{m})^2}},$$

where m_{1k}, m_{2k} are the inversion parameters (normalized resistivity values, the same as in the other terms of the objective function) of the cells of layer k of the two columns, $\bar{m}$ is half the sum of the mean over the first column and the mean over the whole model. The quantity g equals zero when the profiles of the two columns vary with depth in the same way, and grows when they diverge. $w\sum g^2/L$ (L is the number of pairs) is added to the objective function, and a linearized term is added to the system of equations of each iteration: $w\,D^T D$ is added to the matrix and $-w\,D^T g$ to the right-hand side, where D are the derivatives of g with respect to the model parameters (computed analytically at each iteration). Cells with a fixed value (not included in the inversion parameters) are not included in the comparison.

> **Tip.** Start with a small **Pushing factor** (1) and increase it only if the model remains "patchy"; monitor the misfit, which grows if the weight is too large. A large window (**Window size** 4–5) levels the model more strongly but increases the time of each iteration.

RMS vs iteration window

The **RMS vs iteration** window shows the inversion progress: the horizontal axis is the iteration number, the vertical axis the misfit (%). The window opens automatically in the upper right corner of the main window after the first iteration if **Options / Inversion / Show progress** is enabled, and stays on top of other windows; the window can be resized. The plot is cleared at each new start of the inversion. Turning off **Show progress** closes the window.

The plot is convenient for assessing convergence:

- a fast drop of the misfit in the first 2–4 iterations followed by a "plateau" is normal convergence; further iterations give little;
- a slow monotonic decrease: the inversion can be continued or Smoothing factor decreased;
- growth or oscillation of the misfit is a sign of instability: increase Smoothing factor, enable Robust weighting scheme, check the data for outliers, for Focused increase Threshold;
- a plateau at a high level (more than 10 %): systematic errors in the data or geometry, wrong topography or an insufficient mesh are likely.

Misfit evaluation of inversion results

A quick estimate of the result quality is given by the relative misfit **RMS, %**, shown in the status bar of the main window and on the **RMS vs iteration** plot. It is computed over all measurements for which calculated values exist.

As a rule, with satisfactory data quality the misfit should not exceed 5 %; for good-quality data 1–3 % is usually achieved. Too small a misfit (below the noise level) is a sign of "fitting the noise": the model may contain false details. In this case increase Smoothing factor.

The misfit of each measurement can be assessed on the contour map or pseudosection by selecting **Options / Data / Data misfit**. Clusters of large misfits in one area point to problems with particular stations, loop geometry or an insufficiently detailed model in that place; single large misfits point to outliers.

A priori information

A priori information is everything known about the section besides the measurements themselves: geological sections and maps, seismic profiles, drilling and logging data, models of previous interpretations. The program uses such information in two ways:

- as a graphic background: it helps compare the obtained model with known geology but does not affect the inversion;
- as numerical information: models and borehole data that are built into the starting model or overlaid on sections for control.

All commands for working with a priori information are in the **Options** menu of the main window: the **Import/Export** submenu (backgrounds and model build-in) and **Boreholes** (borehole data).

Graphic backgrounds

A background is a raster image georeferenced to the model coordinates and displayed under it (or together with it). Typical examples are a scan of a geological section, a seismic time section, a topographic map or a satellite image. The program supports two kinds of backgrounds: 2D for vertical slices and 3D for the volume window.

2D background

A 2D background is bound to a particular vertical slice. Each slice (XZ or YZ direction and slice number) can have its own image; when moving to another slice the program shows its background if one is set. For horizontal slices (XY) a 2D background is not loaded; in this case the command does nothing.

How to load:

1. Select the required vertical slice of the model.
2. Select **Options / Import/Export / Background 2D / Load background**.
3. Specify an image file (*.bmp, *.png), a seismic file (*.sgy) or a text georeference file (*.sec).
4. For raster files a coordinate dialog opens: the coordinates of the upper left and lower right corners of the image (X1, Z1, X2, Z2) in model units. The **Insert topography** option bends the image along the model surface topography; it is used for sections built from the topography line.

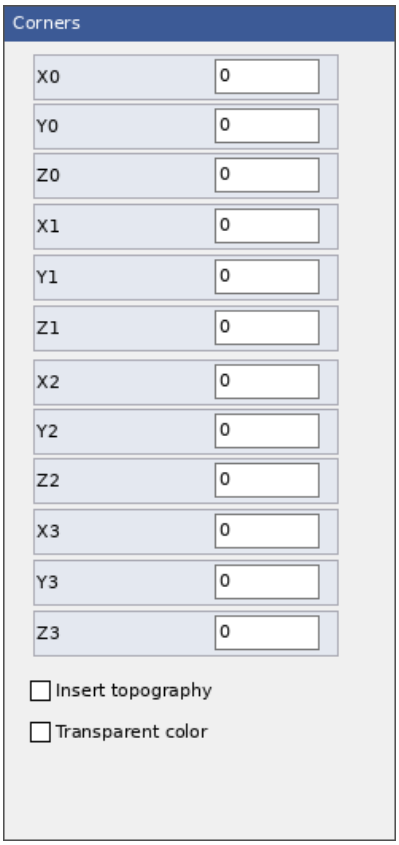
The **Options / Import/Export / Background 2D / Remove current** command removes the background of the current slice; backgrounds of the other slices are kept.

The *.sec file is convenient if the same background has to be loaded many times: the image path and corner coordinates are written in it once, and the coordinate dialog is not needed. The format is described in the “File formats” appendix.

> **Tip.** If the background corner coordinates are known in kilometres, it is enough to add the word km to the *.sec file: the values will be multiplied by 1000 automatically.

3D background and satellite map

A 3D background is one image placed in the space of the volume window. It is used, for example, to show a geological map on the model surface or a vertical section crossing the model.



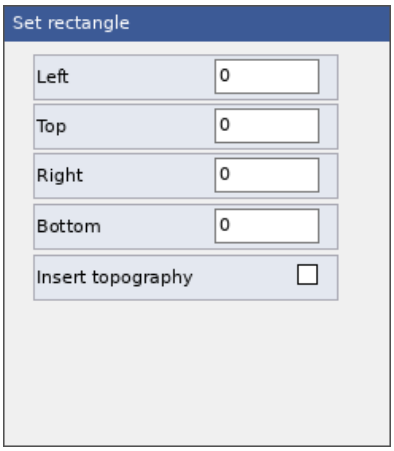
Corners window

- **Options / Import/Export / Background 3D / Load background** loads an image and opens a dialog in which the spatial coordinates of the four image corners (X, Y, Z of each corner) are set. By default the coordinates of the current slice are used.
- **Load satellite map** loads a georeferenced satellite image or map; the program asks for the UTM zone and places the image on the top horizontal plane of the model.
- **Remove background** removes the 3D background.

Background visibility in the volume window is toggled with the **Show / Background** item of the 3D view menu (right click in the 3D window).

Model build-in

Build-in transfers to the current model a parameter distribution from another source: from the results of a previous 3D inversion, from a set of 2D sections or from an arbitrary point cloud (for example, logging or seismic tomography data converted to resistivity). The built-in model then serves as the starting model for inversion or is used for comparison.



Set rectangle window

Options / Import/Export / Build-in mod3d opens a file whose type is selected by the dialog filter:

Filter	What happens
MOD3D model	Values from the model file are interpolated into the current mesh; the project mesh does not change
MOD3D mesh&mod	The project mesh and model are completely replaced by the mesh and model from the file
XYZV	Values of the current parameter are interpolated from the X, Y, Z, V point cloud

Interpolation uses the 12 nearest points with inverse-distance weights; for parameters with a logarithmic scale (for example, resistivity) logarithms are averaged. Values are overwritten in all model cells, so if the current model must be kept, it should first be placed in the buffer (**Buffer** menu).

Build-in mod2d's builds in one or several 2D models (*.mod files from 2D interpretation programs). Several files can be selected at once; the points of all sections are combined into a common cloud and interpolated into the whole 3D model mesh in the same way.

> **Attention.** An XYZV file needs at least six points. Lines with fewer than four numbers are skipped, so the file may contain header and comment lines.

Borehole data

Borehole data (lithological columns and logging curves) are the most accurate a priori information. The program displays them on vertical model slices and in the volume window, which allows checking how well the model boundaries correspond to the boundaries penetrated by drilling.

Commands of the **Options / Boreholes** submenu:

Item	Purpose
Create/Edit borehole data	Opens the borehole editor window
Load borehole data	Loads boreholes from a *.crt file or 1D models from a *.mod1d file
Remove borehole data	Removes all borehole data from the project
Set column's width	Column width on the section as a percentage of the image size (from 0.05 to 5)

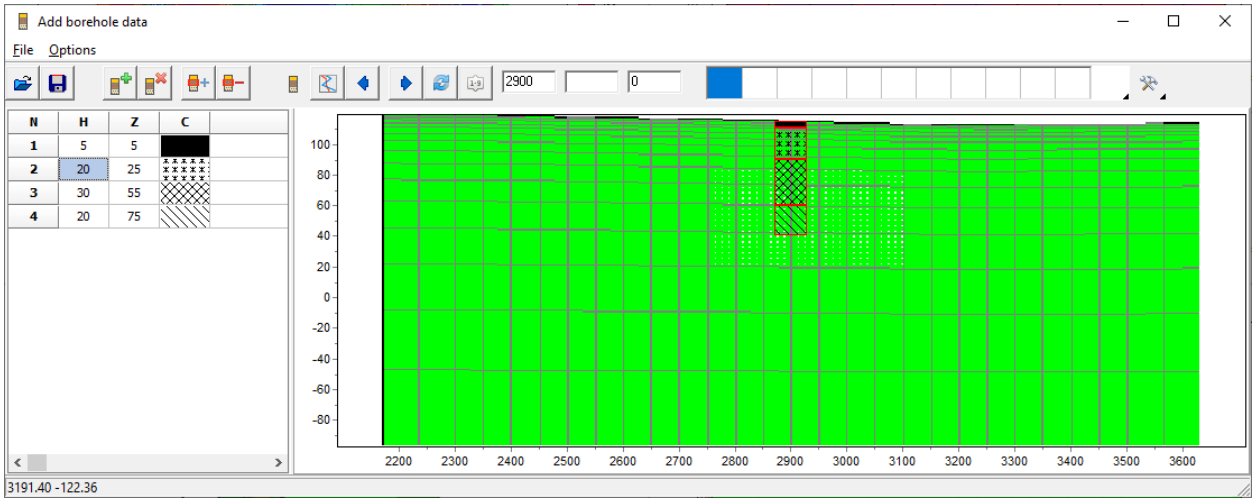
Item	Purpose
Edit positions	Table with names and X, Y coordinates of all boreholes for manual editing

How boreholes are bound to the model

Each borehole has collar coordinates X, Y, collar elevation and inclination angle. A vertical slice shows the boreholes located closest to the slice plane; the position along the slice is recalculated from the collar coordinates. Columns are not shown on horizontal slices (XY). An inclined borehole is drawn in the slice plane at the given angle. The column width is set with the **Set column's width** item and is not related to the real diameter.

Borehole editor window

The window is opened with the **Options / Boreholes / Create/Edit borehole data** item. On the left is the layer table of the current borehole, on the right a graphic window with all boreholes, at the top a toolbar.



Borehole editor window

Toolbar buttons:

Button	Name	Action
	Open	Open a *.crt borehole file or a *.sec graphic background (the type is selected by the dialog filter)
	Save	Save boreholes to a *.crt file or borehole images to *.bmp files (one file per borehole)
	Add new borehole	Create a new borehole
	Remove borehole	Delete the current borehole (the last remaining one cannot be deleted)
	Add layer	Add a layer after the selected row
	Delete layer	Delete the selected layer
	Lithological column mode	Lithological column mode
	Logging mode	Logging curve mode (loading

Button	Name	Action
		logging data)
—	Previous	Go to the previous borehole
—	Next	Go to the next borehole
	Repaint	Redraw the graphic window after editing the table
	Sort by coordinates	Sort boreholes by horizontal coordinate
	Options	Menu: Load default palette, Save default palette, Set borehole width

Toolbar fields:

Field	Purpose
X	Horizontal coordinate of the borehole along the slice
Title	Borehole label (name)
Angle	Borehole inclination angle, degrees

The background colour button on the panel changes the colour of the editor graphic window.

File (Open project, Save project, Exit) and **Options** (Add borehole, Remove borehole, Add layer, Remove layer, Prev, Next, Refresh image, Sort boreholes, Remove background, Show background) window menus duplicate the toolbar buttons. A background (*.sec) opened in the editor allows digitizing boreholes directly on a scanned section.

Layer table

Column	Contents
N	Layer number
H	Layer thickness
Z	Depth of the layer bottom
C	Layer colour and hatching
fourth column	Text label of the layer (rock name); the column header is editable

Thicknesses and depths are entered in section axis units. Columns H and Z are linked: when a thickness changes, the depths of the underlying layers are recalculated, and vice versa. The bottom depth must lie between the depths of the neighbouring layers.

To enter a column quickly from an Excel table, create the required number of layers, copy the thickness column and paste it into column H with Ctrl+V. Values are pasted only into existing rows.

Layer colour and hatching:

- a repeated left click on cell C of the selected row opens the hatching and colour editor;
- a right click on cell C opens the exchange menu: **Exchange P&C** (apply the hatching and colour of the current layer), **Exchange P** (hatching only), **Exchange C** (colour only) to all layers of the same lithology in all boreholes; layers with the same hatching (for **P**) or with the same hatching and colour (for **C** and **P&C**) are considered identical;

- a right click on a layer in the graphic window also opens the hatching editor.

Lithology palette

Below the table there is a palette of 11 cells, a set of frequently used “colour + hatching” combinations. A left click on a cell applies it to the current layer, a right click stores the colour and hatching of the current layer in the cell. The palette is reset at each start, so a customized palette is saved with the **Options / Save default palette** command (button) and loaded with the **Load default palette** command.

Creating a lithological column

1. Press (Add new borehole).
2. Enter the horizontal coordinate, label and inclination angle in the panel fields.
3. Use the button to create the required number of layers. A new layer is inserted after the selected row with zero thickness and the colour of the selected layer.
4. Enter thicknesses (H) or bottom depths (Z), assign colour and hatching.
5. Press : the borehole appears in the graphic window.
6. Save the boreholes with the button to a *.crt file.

Loading logging data

A logging curve is the dependence of a measured quantity (for example, apparent resistivity or natural gamma activity) on depth. On the section the curve is drawn in a strip of column width next to the borehole axis.

How to load:

1. Create a borehole (Add new borehole) or select an existing one.
2. Press (Logging mode) and select a file.
3. For a *.txt or *.dat text file, lines containing at least two numbers are used: the first is depth, the second the value. At least four such lines are required.
4. For a *.las file a column selection window opens (see below).
5. Set the curve display parameters in the **Settings** window.

Settings window (logging parameters)

Element	Purpose
To logarithm	Display logarithms of the values (for resistivity and other quantities varying by orders of magnitude)
Type of norming	Curve normalization in the column strip: common limits, common limits for all curves with the same data index; common limits/med, common limits after subtracting the median of each curve; med, median subtraction
Data index	Data type index (0–5). Curves with the same index are normalized together; different indices allow showing, for example, resistivity and gamma logs independently
Color	Curve colour
Title	Curve name

Element	Purpose
Depth units	Depth units in the file: m, km, ft
Close	Apply and close the window

Normalization is needed because the column width on the section is small and fixed: the curve is stretched over the whole strip width. With common normalization the curve amplitudes of different boreholes are comparable; median subtraction allows comparing curve shapes when their levels differ strongly.

> **Attention.** If the To logarithm option is on, the curve values must be positive.

LAS file column selection window

A LAS file contains several curves in columns. After opening, the program replaces empty values (NULL) and shows the data table.

Element	Purpose
Row 1 of the table	Column type: par1, par2, par3 (curves to load) or Z (depth)
Row 2 of the table	Units: lin or log for parameters; m, km or ft for depth
Start	Mark the selected row as the first data row
End	Mark the selected row as the last data row
OK	Load the selected columns

Each selected parameter is loaded as a separate curve at the same point (the curve name is composed of the borehole name and the data type). Depths are taken by absolute value, so the sign of depth in the file does not matter.

Loading 1D models

The **Options / Boreholes / Load borehole data** item with the *.mod1d filter loads 1D interpretation results (for example, from ZondTEM1D) as columns coloured by parameter value. Each file block is one point with coordinates and a set of layers. Such columns are convenient to compare with the 3D model: layers are drawn with the colours of the same scale as the model. Layers of such columns are not editable, and the mode buttons are unavailable for them. The format is described in the “File formats” appendix.

Boreholes in the volume window

In the volume window boreholes are shown as vertical (or inclined) columns at the collar points. Borehole visibility is toggled with the **Show / Boreholes** item of the 3D view menu.

Mouse and keys in the borehole editor

Action	Combination	Where
Paste the thickness column from the clipboard	Ctrl+V	Column H of the table
Open the hatching and colour editor	Repeated left click on the selected row	Column C of the table
Menu Exchange P&C / P / C	Right click	Column C of the table
Select borehole and layer	Left click	Graphic window

Action	Combination	Where
Layer hatching editor	Right click on a layer	Graphic window
Apply a palette cell to the layer	Left click	Palette
Store the layer in a palette cell	Right click	Palette

Interpretation results

The inversion result is a 3D distribution of parameters (resistivity) in the mesh cells. The program shows the model simultaneously in two views: on a plane slice (section window) and in the volume window. The plane slice is convenient for detailed analysis and comparison with boreholes, the volume window for understanding the shape of anomalies in space.

Model slice

The section window shows one model slice, vertical (XZ or YZ) or horizontal (XY). The slice direction is selected in the **Options / Model / Plane** menu (**Alt+1/2/3**), the number with the  buttons; rotating the mouse wheel over the section switches to the previous or next slice of the same direction.

The **Options / Model / Block-section** / **Smooth section** menu item switches the slice view: in block mode each cell is filled with one colour, in smooth mode the colour is interpolated between nodes.

Volume window

The volume window shows the model in 3D perspective. Mouse control:

Action	Combination	Where
Zoom in/out	Mouse wheel	Volume window
Rotate the model	Moving with the left button pressed	Volume window
Pan the image	Moving with the middle button pressed	Volume window
Open the 3D settings window	Right click	Volume window
Switch slice	Mouse wheel	Section window

3D view menu

The  button on the toolbar opens the volume window menu:

Item	Purpose
3D view settings	Open the 3D settings window
Print preview	Print preview and printing of the image
Plane cut	Selection of the slice plane direction: XZ, YZ, XY
Model / Halfspace value	Set a homogeneous half-space value or a vertical gradient (minimum and maximum) and fill the model with it
Model / Clear model	Clear the model (return the starting value)
Model / Clear limits	Remove parameter value limits in cells
Model / MinMax auto	Automatically set colour scale limits
Model / Smooth/raster	Smoothing or rasterization of the model (see below)
Axises settings	X, Y, Z axis settings

Item	Purpose
Show	Display of elements: Cutting plane (plane of the current slice), Survey points (stations and loops), Boreholes, Background (3D background)
Synchronous cut	The slice in the section window follows the position of the cutting plane in the 3D settings window
X:Y:Z=1:1:1	Equal scale on all axes

> **Attention.** The Halfspace value command replaces the current model with a half-space or gradient. If the model is needed, it should first be placed in the buffer or saved.

The  button inserts a slice copied (with the  button) into the model: a left click inserts the slice, a right click opens a menu for choosing the insertion range: **Current** (current slice only), **Current&+** (from the current to the last), **Current&-** (from the first to the current). This allows, for example, setting a layer persistent along strike in the starting model by drawing it on one slice.

3D settings window

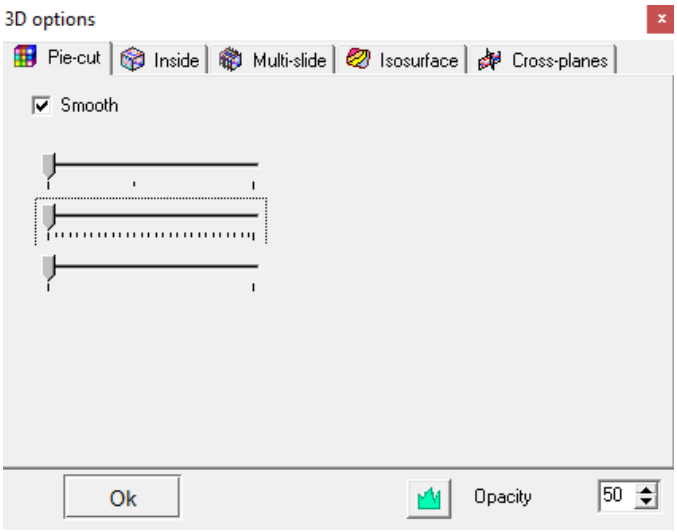
The window is opened with a right click on the volume window or with the **3D view settings** item. The window tabs set the visualization mode, i.e. the way the volume is turned into an image. Selecting a tab immediately redraws the volume window.

Common window elements:

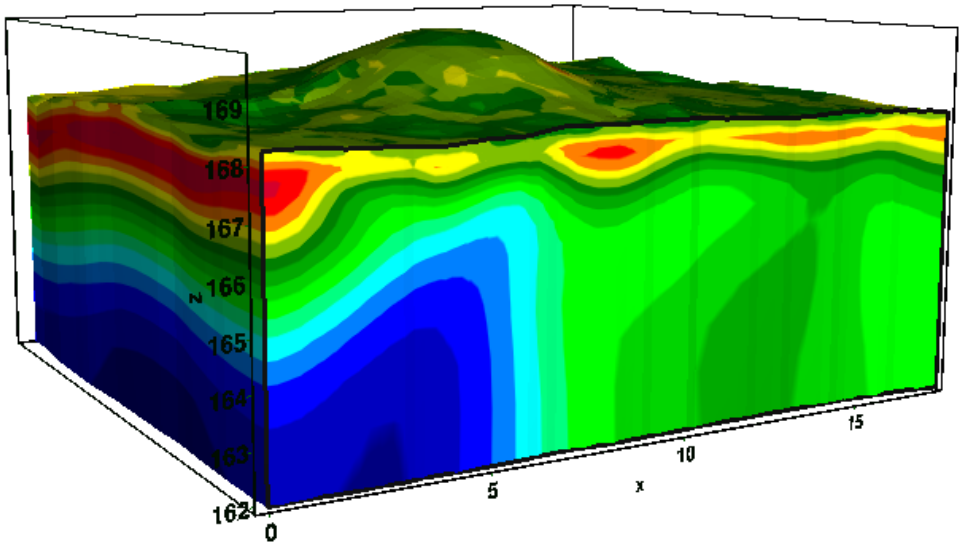
Element	Purpose
Opacity	Model opacity, 0–100 % (50 by default); sets the same opacity for all parameter values
Opacity vs parameter	Window for setting opacity depending on the parameter value
Smooth	Image type: off, each cell has its own colour; on, the colour is interpolated between nodes. The Smooth check boxes on different tabs are linked
Ok	Save the settings and close the window

Pie-cut

The model is shown as a solid block from which a corner sector is cut out. Three sliders set the position of the cutting planes in X, Y and Z; the part of the model between the model corner and the planes is cut out. The mode is convenient for viewing the internal structure while showing the model faces.



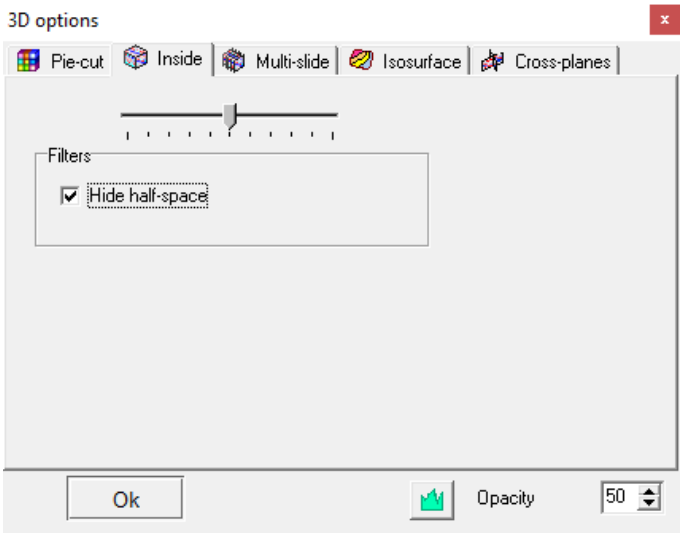
3D settings window, Pie-cut tab



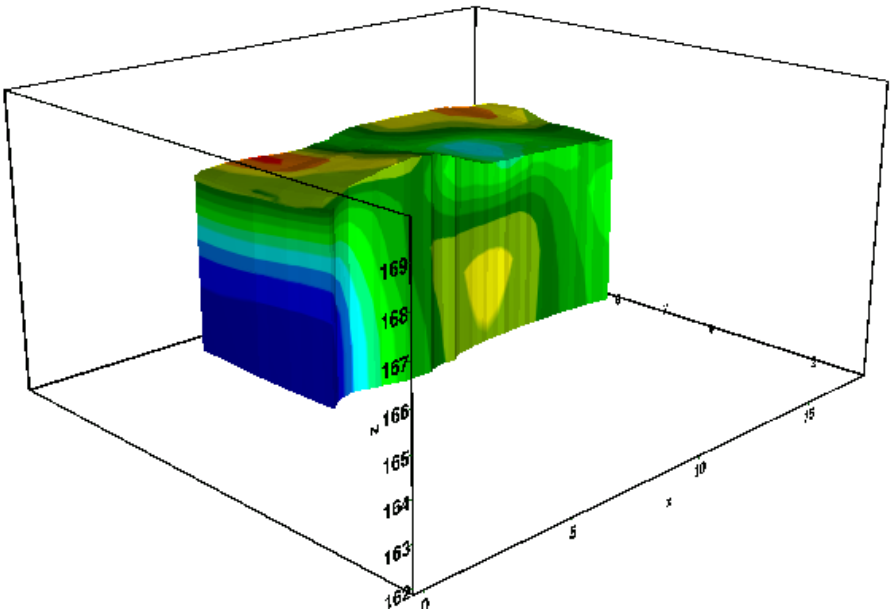
Example of the Pie-cut view

Inside

The model is shown as a set of separate cubes at the cell centres. The slider sets the cube size as a percentage of the cell size: decreasing the size makes the model “transparent” and allows seeing internal anomalies. Filters:



3D settings window, Inside tab

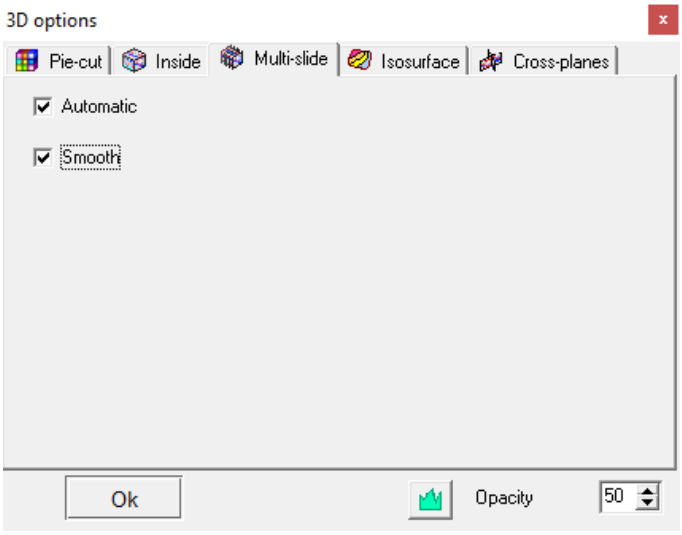


Example of the Inside view

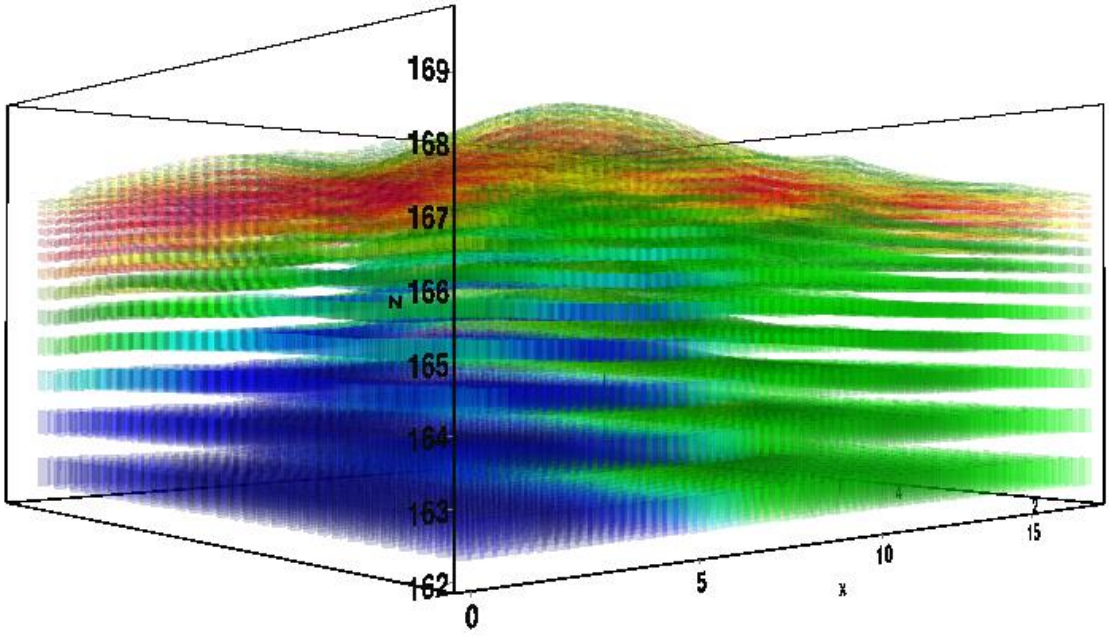
Element	Purpose
Hide half-space	Do not show cells whose value equals the background value (on by default)
Cutted by planes	Show only cells not cut off by the slice planes

Multi-slide

The model is shown as a set of parallel semi-transparent slices. The **Automatic** check box selects the slice direction depending on the view angle, so the image remains readable during rotation. The mode is close to volume rendering and shows the shape of anomalies well in combination with the Opacity vs parameter setting.



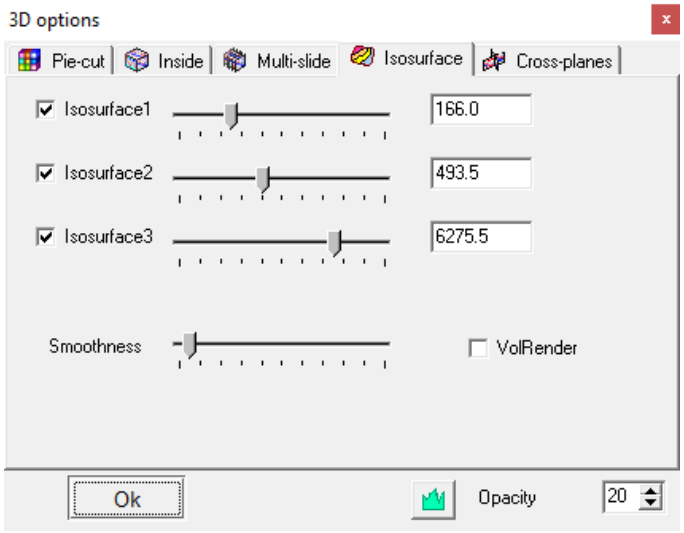
3D settings window, Multi-slide tab



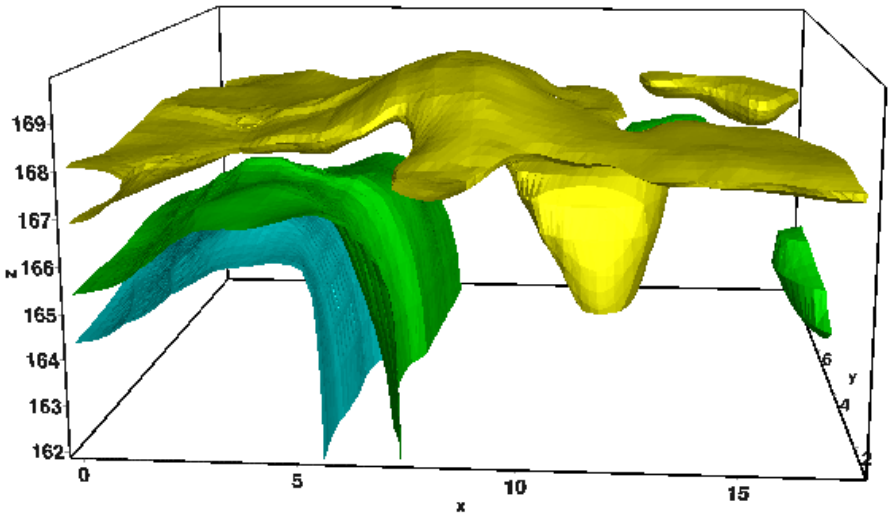
Example of the Multi-slide view

Isosurface

Isosurfaces (surfaces of equal parameter value) are built. Up to three isosurfaces can be shown simultaneously.



3D settings window, Isosurface tab



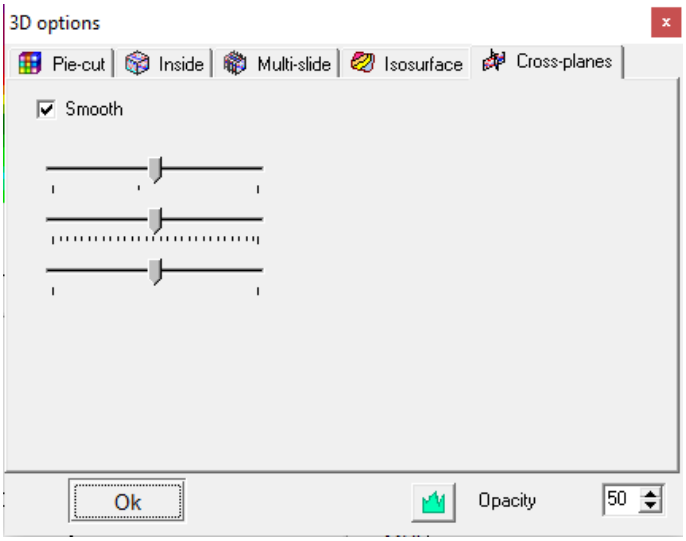
Example of isosurfaces

Element	Purpose
Isosurface1, Isosurface2, Isosurface3	Enable the isosurface
Sliders	Level position as a percentage of the colour scale range (for logarithmic parameters, on a logarithmic scale)
Value fields	Level value in parameter units; the slider and field are linked
Smoothness	Degree of surface smoothing
VolRender	Additionally show the model as a semi-transparent “fog”

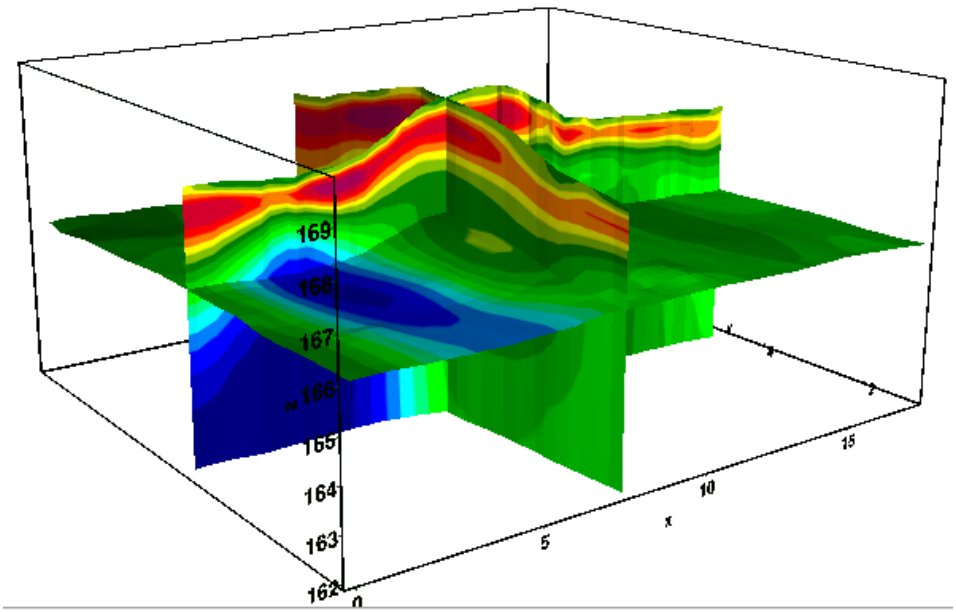
Isosurfaces are the most illustrative way to show the outline of a conductive or resistive body. The level value is chosen so that it separates the anomaly from the host medium.

Cross-planes

Three mutually perpendicular slices (XZ, YZ, XY) are shown, their positions set by three sliders. If the **Synchronous cut** option is on, the slice in the section window follows the slider of the current direction.



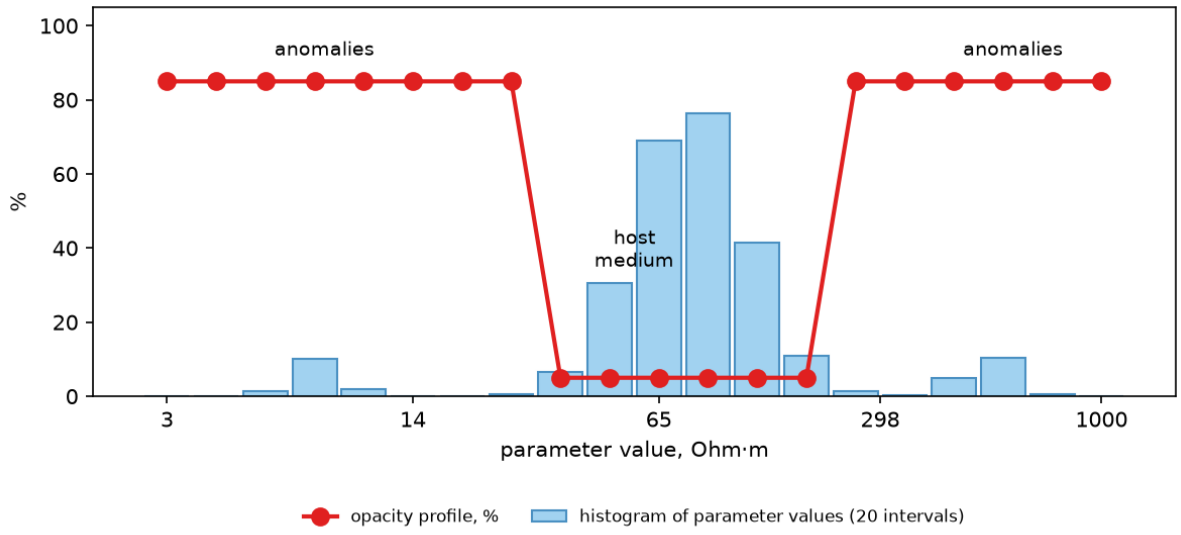
3D settings window, Cross-planes tab



Example of the Cross-planes view

Opacity depending on the parameter

The **Opacity vs parameter** window (button in the 3D settings window) allows making some parameter values transparent. For example, to see conductive bodies in a resistive sequence, high resistivities are given zero opacity and low ones the maximum.



Parameter histogram and opacity profile

The window shows a histogram of the current parameter values (20 intervals over the colour scale range) with the colour scale and a red line of the opacity profile of 20 points. Opacity values range from 0 to 100. The profile is set separately for each parameter.

Action	Combination	Where
Change opacity at a point	Dragging a point with the left button	Opacity vs parameter window
Set the same value for all points to the left (when dragging a point in the left half) or to the right (in the right half)	Ctrl + drag	Opacity vs parameter window
Set the level for all points	Right click	Opacity vs parameter window

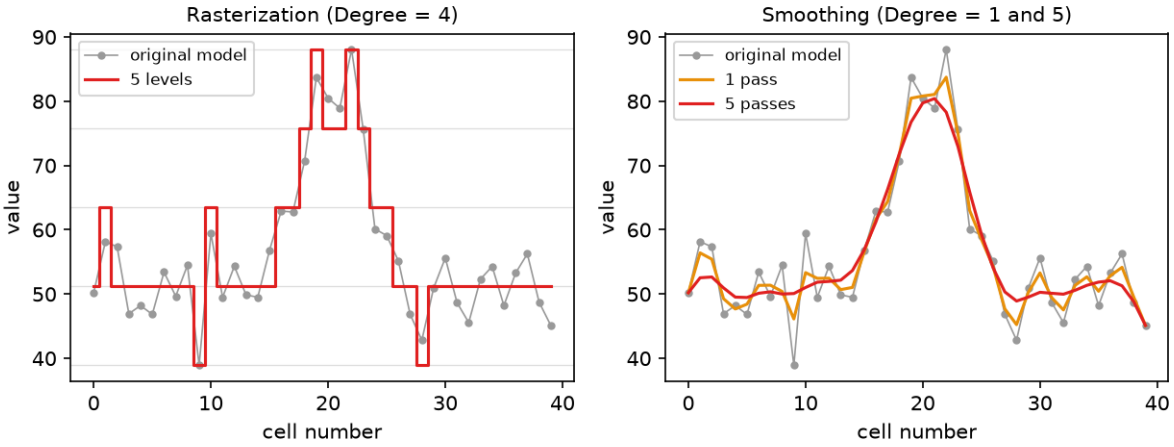
The model in the volume window is redrawn after the mouse button is released. The Opacity field in the 3D settings window resets the whole profile to one value.

Model smoothing and rasterization

The **Smooth/raster model** window (**Model / Smooth/raster** item of the 3D view menu) transforms the current model:



Smooth/raster model window



Rasterization and smoothing on a slice example

- **Rasterization** reduces values to a finite set of levels (the model becomes "blocky"). This is useful for preparing a starting model for inversion with block parameterization and for clearly distinguishing several geological classes;
- **Smoothing** smooths the model, removing small inversion artefacts.

Element	Purpose
Operation	Rasterization or Smoothing
Degree	For Rasterization, the number of intervals (there is one more level); for Smoothing, the number of smoothing passes. From 1 to 20, 5 by default
Apply	Save the result and close the window

In rasterization the value range is divided into Degree equal intervals $\Delta = (max - min)/Degree$ (for logarithmic parameters, in logarithms), and the value of each cell is replaced with the nearest level. In smoothing, in one pass the cell value is replaced with a weighted average: the cell with weight 3 and its six neighbours with weight 1:

$$v' = \frac{3v_0 + \sum_{k=1}^{6} v_k}{9},$$

for logarithmic parameters the geometric mean is used.

The result is recalculated from the original model at each parameter change and shown in the windows immediately. Closing the window without **Apply** returns the original model. Only the current parameter is transformed; value limits in cells are not taken into account.

Model cutting

Cutting (blanking) hides parts of the model not covered by data, i.e. mesh edges outside the survey area. These cells remain in the model but are not shown on sections and in the volume window and are not written to XYZV/XYZ export files. The commands are in the **Options / Cutting** menu:

Item	Purpose
Survey contour	Cutting along the survey station contour
User polygon	Cutting along a user-defined plan contour
Groups cutted off cells	Combine cut-off cells with the nearest non-cut cells in the inversion

Item	Purpose
Blanking cut	Turn cutting display on or off

Cutting along the data contour

Survey contour builds the cutting boundary along the outer contour of the stations: cells outside the survey area are hidden. This is the most natural way to show only the part of the model covered by data.

Cutting by polygon

User polygon opens the **Set contour of cutting** window, in which a contour is set on the station plan. After the contour is applied, cutting turns on automatically. A cell is kept if more than four of its nine control points (corners, side midpoints and centre) fall inside the contour; cutting acts at all depths.

Button	Purpose
Add user defined polygon	draw a contour with the mouse
Create polygon automatically	build a contour along the outer boundary of the stations
Remove polygon	delete the contour
Apply polygon	apply the contour and close the window

Action	Combination	Where
Add a contour vertex	Left click	Set contour of cutting window, drawing mode
Close the contour	Right click (at least two vertices)	Set contour of cutting window, drawing mode

Plot zooming is disabled while drawing a contour.

Combining cut-off cells

If **Groups cutted off cells** is on, each cut-off cell receives in the inversion the parameter of the nearest non-cut cell of the same layer. The number of independent unknowns decreases, which speeds up the inversion and makes the model edges more stable.

Several models in a project

The model buffer allows keeping up to four models in a project simultaneously, for example inversion results with different smoothing parameters or a model before and after building in a priori information, and switching between them quickly.

The **Buffer** menu contains cells **Model 1**–**Model 4**:

- a click on an empty cell puts the current model into it;
- a click on a filled cell opens the **Working with buffer** dialog with the buttons **To buffer** (write the current model to the cell instead of the previous one) and **From buffer** (replace the current model with the model from the cell).

A cell stores model values and limits. A model can be retrieved only into a mesh of the same size.

The **Open** item of the buffer menu shows the **Sections from buffer** window: the current slice of all stored models in a common colour scale for visual comparison.

Saving results

Results are saved with the **File / Save file** command or the  button. The file type is selected by the dialog filter:

Filter	What is saved
Project file	Project: data, mesh, model, settings (*.t3d)
Project with calculated	Project together with calculated data (*.t3d); when such a project is opened, calculated data and RMS are available immediately
Data table [*.txt]	Table of observed and calculated data for all receivers and times (format in the appendix)
XYZV [*.dat]	The whole model at mesh nodes: X, Y, Z and parameter values
XYZ [*.dat]	Current slice at nodes: two coordinates and values

The XYZV and XYZ formats are intended for building maps and volume images in other programs (Surfer, Voxler, Oasis montaj, ParaView, etc.). Values are written at mesh nodes: for each node the average over the eight surrounding cells is taken (for a slice, over four), for logarithmic parameters the geometric mean. Cells hidden by cutting are not written when **Blanking cut** is on. The columns are described in the “File formats” appendix.

Other export commands (Export mod3d, Export mod2d, Export to Paraview/Oasis, Draw to voxler, images and video) are in the **Options / Import/Export** menu and are listed in the “Main menu” chapter.

Building slices in Surfer

The **Options / Import/Export / Draw to surfer** command builds maps of horizontal model slices at given depths in Surfer. Before building, the **Set depths** window opens.

Element	Purpose
X grid number	Number of grid nodes in X (10–10000, 100 by default)
Y grid number	Number of grid nodes in Y (10–10000, 100 by default)
XY slice number	Number of slices (1–6, 3 by default); when changed, the depth table is filled uniformly over the model depth
# slice / Depth table	Slice numbers and their depths; depths can be edited
3D	On: slices are placed one under another in axonometry; off: as separate maps on one sheet
From surface	Depth is measured from the topographic surface; off: absolute elevations
Build	Build

Depths outside the model are ignored. The table is refilled each time the window is opened.

> **FAQ.** Why have previously built maps changed after a repeated build? The intermediate grid is written to a temporary file that is overwritten at each build. Finished maps should be saved in Surfer under their own names.

Workflow

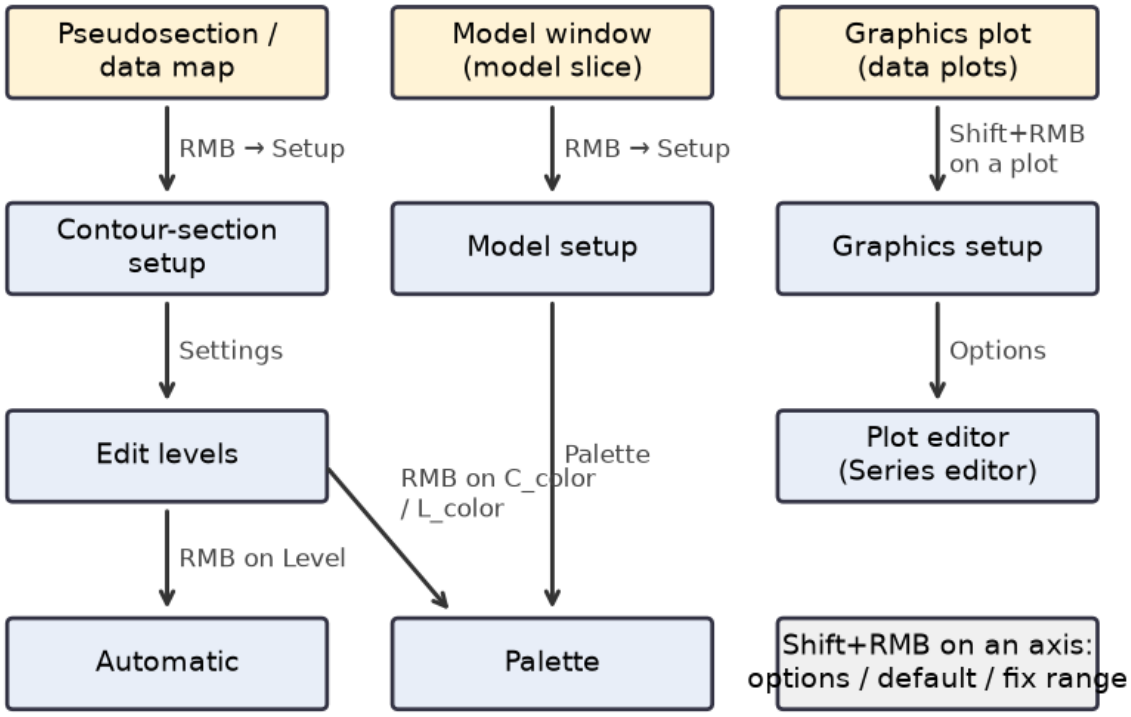
1. Before the inversion, load backgrounds: **Background 2D** on key slices (geological sections, seismics), **Background 3D** or **Load satellite map** for the site map.
2. Create or load boreholes: lithological columns in the borehole editor, logging from *.txt, *.dat or *.las, 1D models from *.mod1d. Save the boreholes to *.crt.
3. If previous results exist, build them into the starting model (**Build-in mod3d**, **Build-in mod2d's**). Put the original model into the buffer first.
4. Set cutting: **Survey contour** or **User polygon**, turn on **Blanking cut**. To speed up the inversion, turn on **Groups cutted off cells**.
5. Run the inversion; store result variants in the buffer (**Model 1–4**) and compare them in the **Sections from buffer** window.
6. Analyse the model on slices (with borehole columns) and in the volume window, choosing the mode in the **3D settings** window and the opacity in the **Opacity vs parameter** window.
7. If necessary, smooth or rasterize the model (**Smooth/raster**).
8. Save the project (*.t3d) and export the model to XYZV or slices to XYZ; build slice maps in Surfer (**Draw to surfer**).

Mouse and keys of this chapter

Action	Combination	Where
Zoom in/out	Mouse wheel	Volume window
Rotate the model	Left button + drag	Volume window
Pan the image	Middle button + drag	Volume window
Open the 3D settings window	Right click	Volume window
Previous/next slice	Mouse wheel	Section window
Insert the copied slice	Left click	Slice paste button
Select the slice insertion range	Right click	Slice paste button
Change opacity at a point	Point dragging	Opacity vs parameter
Set a value for all points to the left/right	Ctrl + drag	Opacity vs parameter
Set the level for all points	Right click	Opacity vs parameter
Add a contour vertex	Left click	Set contour of cutting
Close the contour	Right click	Set contour of cutting
Paste thicknesses	Ctrl+V	Borehole editor, column H
Hatching and colour editor	Repeated left click	Borehole editor, column C
Menu Exchange P&C / P / C	Right click	Borehole editor, column C
Layer hatching editor	Right click on a layer	Borehole editor, graphic window
Apply / store a palette cell	Left / right click	Borehole editor, palette

Graphic settings

All graphic objects of ZondTEM3D (pseudosections and data maps, curve plots, model slices, axes, palettes) are customized with a set of common dialogs. They do not change the data and model but only affect how the result looks on screen, when printed and in saved images. Colour and level settings are saved together with the project, so a design chosen once carries over to all subsequent sessions with this project.



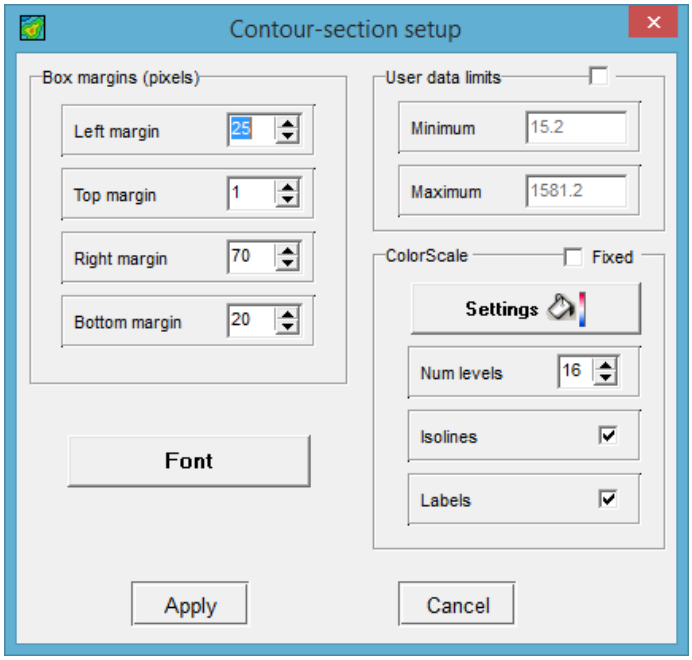
Relationship of graphic settings windows and ways of opening them

The dialogs are linked to each other: an object setup window opens the levels window, the levels window opens the palette, etc. | Object | How to open the setup | Window | |:—|:—|:—| | Pseudosection, data contour map | Right click on the object field outside the plot area → **Setup** | Contour-section setup | | Model slice (model window) | Right mouse button → **Setup** | Model setup | | Curve plots in **Options / Data / Graphics plot** mode | Shift + right click on any curve | Graphics setup | | Individual curve (line, points, labels) | **Options** button in the Graphics setup window | Plot editor | | Axis of any object | Shift + right click on the axis → **options** | Axis editor | | Printing | **File / Print preview** or **Print preview** in the object context menu | Print preview | | 3D model rotation video | **Options / Import/Export / Create video** | Record video |

> **Tip.** The colour scale of the 3D volume model is taken from the model slice palette. To change the colours of the 3D image, adjust the palette in the Model setup window of one of the model slices: the 3D view is redrawn with the same scale.

Contour map and pseudosection setup dialog

The **Contour-section setup** dialog controls the appearance of objects built with isolines and colour fill: pseudosections, maps of apparent resistivity or signal for a selected time, data maps. It is needed when the automatically chosen scale limits are unfortunate (for example, a few anomalous values “stretch” the scale), when several objects must be compared in the same scale or a figure must be prepared for a report.



Contour-section setup window

The dialog is opened with the **Setup** item of the object context menu. The context menu is called by a right click on the object field outside the plot area (on the margins, near the axes or the colour scale).

Element	Purpose	Range / default
Box margins (pixels): **Left/Top/Right/Bottom margin**	Margins of the plot area from the window edges, pixels. Margins leave room for axis labels and the colour scale	0–100
Num levels	Number of levels (contours) of the colour scale and isolines	2–100
Isolines	Show or hide isoline lines over the colour fill	—
Labels	Label values on isolines. Available when **Isolines** is on; not shown for objects where labels are impossible	—
Settings	Open the **Edit levels** window for manual setting of levels, fill colours and isolines	—
Fixed	Fix the colour scale settings: levels and colours are not recalculated when the data are updated	—
User data limits: check box	Use user-specified scale minimum	Off

Element	Purpose	Range / default
	and maximum instead of those found from the data	
Minimum, **Maximum**	Minimum and maximum scale (isoline) value. Active when the check box is on	—
Font	Font of the colour scale labels	—
Apply	Apply the parameters and redraw the object; the window stays open	—
Cancel	Close the window. Changes already applied with the **Apply** button or made in the **Edit levels** window are kept	—

How it works. When **Apply** is pressed, the program rebuilds the triangular mesh over the data points, forms levels from the **Num levels**, **Minimum**/**Maximum** values and redraws the fill. If the data are displayed on a logarithmic scale, the levels are distributed uniformly in the logarithm of the value. Limits are entered in display units (for example, Ohm·m).

When **Settings** is pressed, the program turns on the user limits mode and the **Fixed** check box: manually set levels must not be reset at the next recalculation. If levels were changed in the **Edit levels** window, after it is closed the **User data limits** check box is turned on, and the **Minimum** and **Maximum** fields are filled with the first and last level.

> **FAQ.** “Why are two pseudosections (or maps for different times) coloured differently with similar values?” By default each object chooses the scale from its own data. For comparison, set the same **Minimum**, **Maximum** and **Num levels** in all objects and turn on **Fixed**, or save the levels to a .lvl file in the **Edit levels** window and load this file into each object.

Edit levels window

The **Edit levels** window is opened with the **Settings** button and allows setting each level and its colours manually. This is needed, for example, to highlight a narrow value interval, to bring the scale to an organization standard or to match the scale of another program (Surfer).

The window contains a table and a panel with buttons for loading and saving levels.

Column	Contents	Mouse actions
#	Level number	—
C_color	Fill colour of the interval. Fill colours are set for intervals between levels, so there is one more of them than levels; in a cell the upper half refers to the interval above the level in the table, the lower half to the one below	Right button in a cell: choose the colour of the corresponding interval (click in the upper or lower half of the cell)
L_color	Isoline colour of the level	Right button in a cell: choose the isoline colour
Level	Level value in display units	Select a cell and type a value
L_visible	Whether to show the isoline of this	Click on the check box

Column	Contents	Mouse actions
	level	

Clicks on column headers perform group operations:

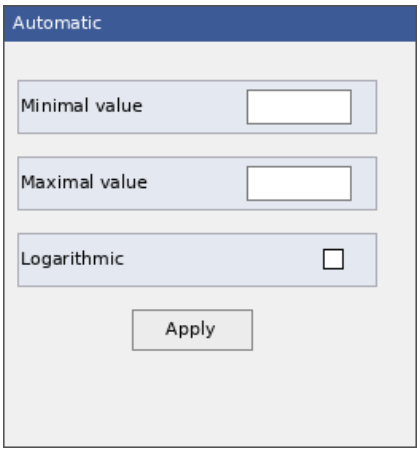
Header	Action	Result
C_color	Right button	Open the fill palette; after the palette is closed, the colours of all intervals are recalculated from it
L_color	Right button	Open the isoline palette; the colours of all isolines are recalculated
Level	Right button	Open the **Automatic** dialog for automatic level placement
L_visible	Left button	Show all isolines
L_visible	Right button	Hide all isolines

Panel buttons: **Load levels from file** loads levels from a .lvl (Surfer LVL2 format) or .clr file; **Save levels to file** saves levels, fill and isoline colours to a .lvl file. The file format is described in the "File formats" appendix.

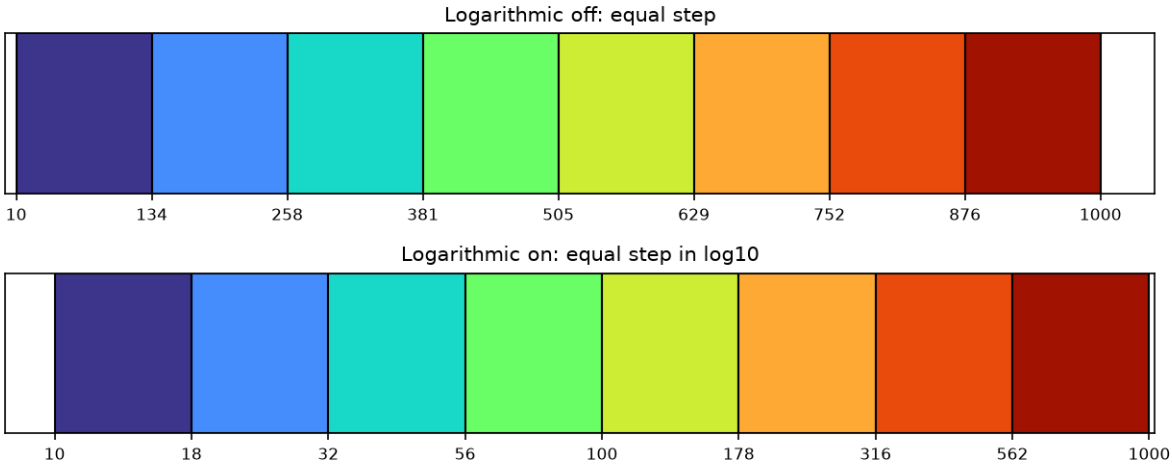
When the palette is opened from the **Edit levels** window, a histogram of data values is displayed above the palette: it is convenient for placing palette sliders where most of the values are concentrated.

Automatic dialog for isoline levels

The **Automatic** dialog is opened with a right click on the **Level** column header in the **Edit levels** window. It distributes the given number of levels uniformly between the minimum and maximum, which is faster than entering each level.



Automatic window



Placement of 9 levels from 10 to 1000 Ohm·m with equal and logarithmic step

Element	Purpose	Default
Minimal value	First level value	Current first level
Maximal value	Last level value	Current last level
Logarithmic	On: equal step in the logarithm of the value; off: equal arithmetic step	Off
Apply	Calculate the levels and close the dialog	—

The number of levels does not change: it is set by the **Num levels** field of the Contour-section setup window. The levels are calculated as follows:

$$L_i = L_{\min} + (L_{\max} - L_{\min})\frac{i}{N-1} \quad \text{or} \quad L_i = exp\left(\ln L_{\min} + (lnL_{\max} - lnL_{\min})\frac{i}{N-1}\right),$$

$i = 0 \ldots N-1$. After calculation each value is rounded to a “round” number: to the order of the smaller of two quantities, the value itself and the step to the neighbouring level. If a level falls between negative and positive neighbours, it is set to zero; this is convenient for sign-changing parameters (for example, a signal changing sign).

The levels do not change if the minimum is not less than the maximum, if a value is entered incorrectly, and also if **Logarithmic** is on and the minimum is not positive.

> **Tip.** For resistivity, which varies by orders of magnitude, almost always use **Logarithmic**: with an equal arithmetic step almost all values will fall into one or two lower intervals.

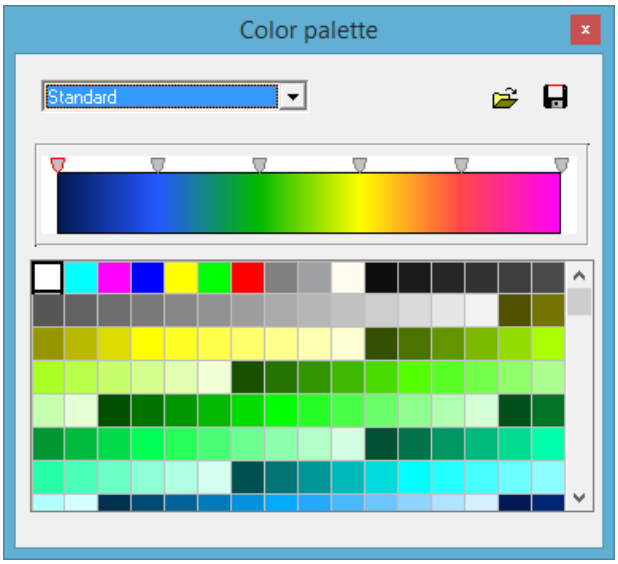
Workflow: a common scale for several objects

1. Open **Setup** in the context menu of the first pseudosection or map.
2. Set **Num levels** (usually 12–20) and press **Settings**.
3. In the **Edit levels** window right-click the **Level** header, enter the minimum and maximum, turn on **Logarithmic**, press **Apply**.
4. If necessary, right-click the **C_color** header and select or edit the palette; the same for **L_color**.
5. Save the levels with the **Save levels to file** button (.lvl) and close the window.
6. In the Contour-section setup window check that **User data limits** and **Fixed** are on, press **Apply**.

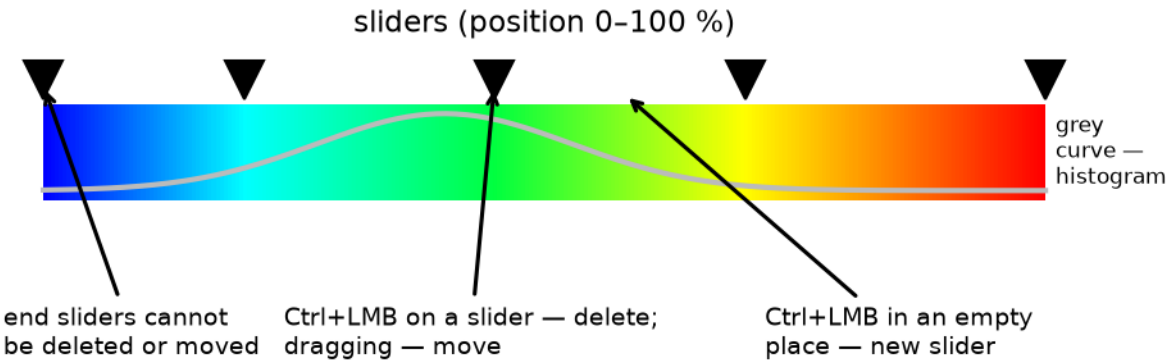
7. In the other objects open **Setup → Settings** and load the saved .lvl file.

Palette setup dialog

The **Palette** window sets the colour scale: fills of pseudosections and maps, isolines, model slices and, through the slices, the 3D model. It is opened with the **Palette** button in the Model setup window or with a right click on the **C_color/L_color** headers in the **Edit levels** window.



Color palette window



Working with palette sliders

The palette consists of sliders, reference colours placed on a scale from 0 to 100 %. Between sliders the colour is interpolated linearly by RGB components (in continuous mode), or the palette is divided into steps (in stepped mode).

Element	Purpose
List of ready palettes	**Rainbow** (blue → green → yellow → red), **Inverse rainbow** (red → yellow → green → blue), **Gray scale** (white → black), **R-W-B** (red → white → blue), **B-W-R** (blue → white → red), **Standard** (dark blue → blue → green → yellow → red → magenta), **Res2dinv** (dark blue → blue → light blue → light green → green → yellow → red → violet), **WinGlink** (red → yellow → light green → light blue → blue → violet). Colours are listed from minimum to maximum values

Element	Purpose
Scale with sliders	Current palette; above it, the reference colour sliders
Grey curve on the scale	Histogram of the object value distribution (if available for the object). Shows in which palette interval most of the values are concentrated
Colour table	A click on a colour assigns it to the selected slider
Continuous	On: smooth (interpolated) palette; off: stepped
N step	Number of steps of the stepped palette, 4–50 (4 by default). The field is shown only in stepped mode
Load	Load a palette from a .csf or .clr file
Save	Save the palette to a .csf file

Slider actions (the mouse must hit the slider zone above the scale):

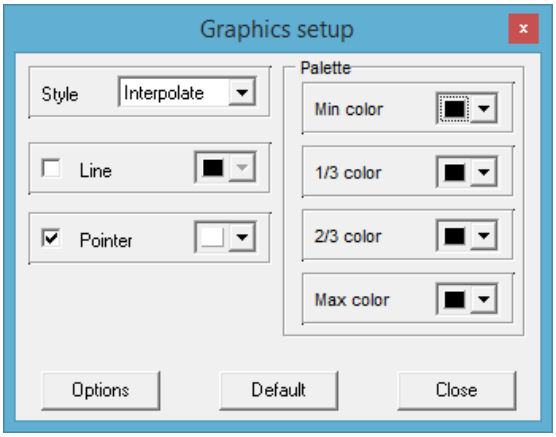
Action	How to do it
Select a slider	Left click on a slider
Move a slider	Press the left button on an inner slider and drag; a slider cannot pass its neighbours
Add a slider	Ctrl + left click in an empty place; the colour of the new slider is interpolated from its neighbours
Delete a slider	Ctrl + left click on an inner slider, or select a slider and press Delete
Change colour	Select a slider and click a colour in the table

The end sliders (0 and 100 %) cannot be deleted or moved, but their colour can be changed.

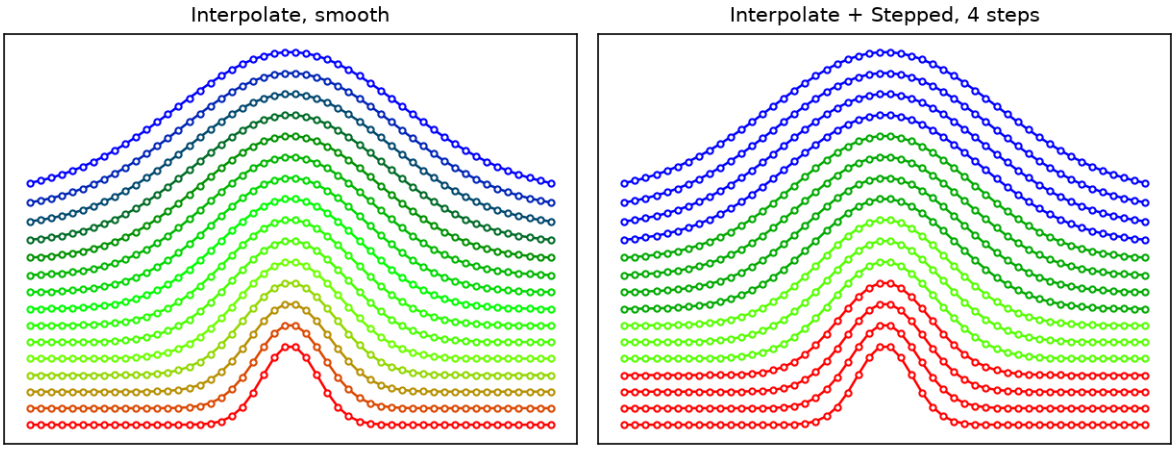
> **Tip.** If the histogram shows that almost all values lie in a narrow interval, move the inner sliders closer to this interval: the contrast in it increases noticeably without changing the scale limits.

Curve set editor

In the **Options / Data / Graphics plot** mode the data are shown as a set of curves (one for each time), observed and calculated separately. The **Graphics setup** window defines how the whole curve set is coloured at once. It is opened with a right click with Shift held on any curve; settings for observed and calculated curves are independent: the window of the set on whose curve the click was made opens.



Graphics setup window



Interpolated and stepped colouring of a curve set

Element	Purpose	Default
Style	Set colouring method: **Interpolate**, colours are interpolated by curve number between **Min color**, **1/3 color**, **2/3 color**, **Max color**; **Constant**, all curves of one colour; **Random**, each curve receives a random colour	Interpolate
Palette: **Min color**, **1/3 color**, **2/3 color**, **Max color**	Reference colours of the interpolated palette for the first, one-third, two-thirds and last curve of the set. In **Constant** mode one colour button is shown instead	Red, light green, green, blue
Line	Check box on: all connecting lines are drawn in the colour selected next to it; off: in the set palette colour	Off
Pointer	Check box on: all points (markers) are filled with the selected colour; off: with the palette colour	On, white
Stepped	Stepped colouring: neighbouring curves are combined into groups of one colour. Available only for **Interpolate**	Off
Numb steps	Number of steps (colour groups), 2–20	4
Options	Open the plot editor for the first curve of the set; the selected line and point style is then applied to all curves of the set	—
Default	Restore the original curve colours and style	—
Close	Apply and close the window	—

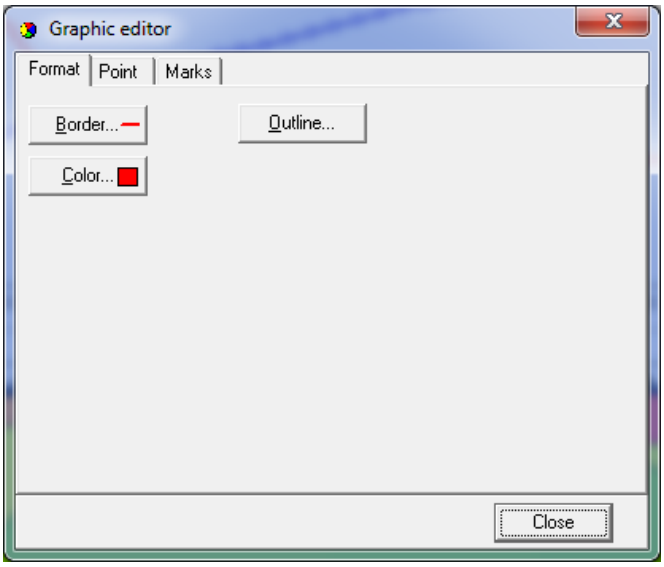
All changes are applied immediately, without a separate confirmation button.

> **Tip.** To compare observed and calculated curves it is convenient to set **Style = Interpolate** with markers without lines for the observed ones, and **Constant** (for example, black) with a line without markers for the calculated ones. Then the fitting discrepancy is visible at once.

A left click on a curve temporarily leaves only it (and the corresponding calculated or observed curve) on the screen and highlights the selected point; after the button is released, all curves are shown.

Plot editor

The plot editor (Series editor) customizes the appearance of an individual curve: line type and colour, point shape and size, labels. It is opened with the **Options** button in the Graphics setup window; the settings are then extended to the whole curve set.



Plot editor window

Format tab: connecting lines:

Element	Purpose
Border	Type, width and colour of the connecting line
Color	Plot colour
Outline	Outline of the connecting line

Point tab: pointers (point markers):

Element	Purpose
Visible	Show or hide pointers
Error gates	Show or hide the confidence interval (error) of values
Style	Pointer shape (circle, square, triangle, etc.)
Width, **Height**	Pointer width and height in screen pixels
Pattern	Pointer fill
Border	Pointer outline line

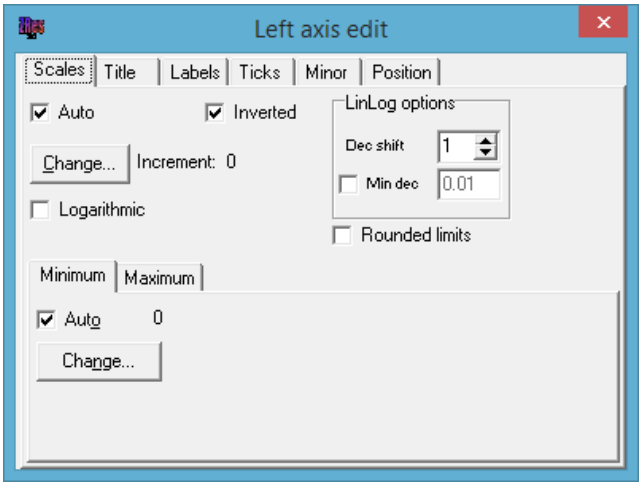
Marks tab: point labels:

Tab / element	Purpose

Tab / element	Purpose
Style: **Visible**	Show or hide labels
Style: **Draw every**	Draw every second, third, etc. label
Style: **Angle**	Rotation angle of the label text
Style: **Clipped**	Do not draw labels extending beyond the plot area
Arrows: **Border**, **Length**, **Distance**	Arrow line from the label to the point, arrow length and distance from its end to the point
Format: **Color**, **Frame**, **Round Frame**, **Transparent**	Label frame background, frame line, rounded corners, frame transparency
Text: **Font**, **Outline**	Label font and letter outline

Axis editor

Most program objects have coordinate axes. The axis editor allows setting limits, logarithmic scale, tick step, labels and the axis title, for example to fix the same depth limits on several slices or to turn on a logarithmic apparent resistivity axis on plots.



Axis editor window (example for the left axis)

A right click with Shift held on an axis opens a menu of three items: **options** opens the axis editor dialog; **default** restores the axis parameters to default values; **fix range** fixes the current axis limits (they stop being recalculated when the object is updated).

Scales tab: axis scale:

Element	Purpose
Auto	Axis limits are determined automatically; with the check box off they are set in the **Minimum** and **Maximum** areas
Inverted	Reverse axis direction (for example, depth downwards)
Increment change	Step of the major axis ticks
Logarithmic	Logarithmic scale. For sign-changing data the **Lin Log options** are used additionally
Lin Log options: **Dec Shift**	Linear-logarithmic axis for data containing zero or values of different signs: offset in logarithmic decades from the largest absolute limit to zero. The decade

Element	Purpose
	nearest to zero is built on a linear scale, the rest on a logarithmic one
Lin Log options: **Min dec**	Set and fix the value of the minimum (near-zero) decade
Rounded limits	Round the axis limits
Minimum, **Maximum**: **Auto**, **Change**	Each limit: automatically or manually with the **Change** button

Title tab: axis title; on the **Style** tab, the text (**Title**), angle (**Angle**), offset (**Size**, 0 means automatic), visibility (**Visible**); on the **Text** tab, the font (**Font**) and letter outline (**Outline**).

Labels tab: tick labels: **Visible**, **Offset** (0 means automatic), **Angle**, **Min separation%** (minimum distance between labels in percent), font and outline on the **Text** tab.

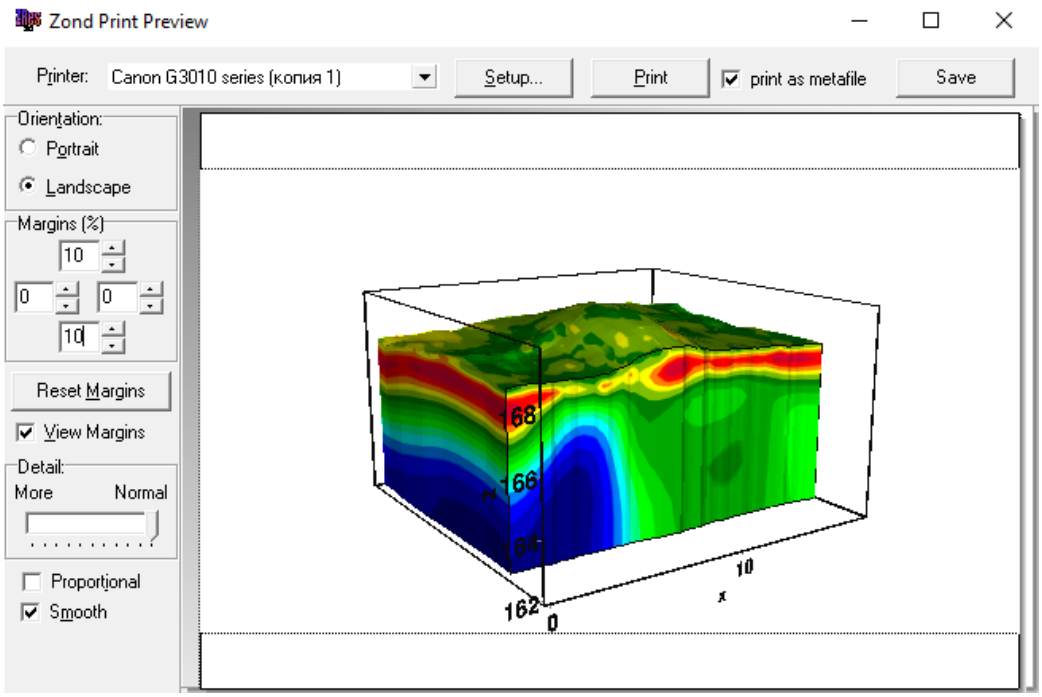
Ticks tab: major ticks: **Axis** (axis line), **Grid** (grid lines), **Ticks** and **Inner** (outer and inner ticks, length set by the **Len** field), **At labels only** (draw ticks only where there is a label).

Minor tab: minor ticks: **Grid**, **Ticks** (length: **Length**), **Count**, the number of minor ticks between major ones.

Position tab: axis position: **Position%** (axis offset relative to the standard position in percent or pixels, depending on **Units**), **Start%** and **End%** (offset of the axis start and end in percent of the plot size).

Print preview dialog

Print preview allows placing objects on the sheet, choosing the printer and paper format, printing the image or saving it to a file.



Print preview window

The **File / Print preview** command puts the data window and the model window (with the 3D image, if shown) together on the sheet. The **Print preview** item in the context menu of an individual object prints only this object.

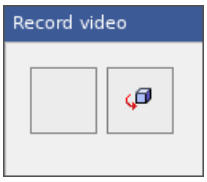
An object is moved on the sheet with the left mouse button. Window panel buttons:

Button	Purpose
Printer	Select a printer from the list configured in the system
Setup	Print setup: paper size and orientation, printer properties, number of pages per sheet
Print	Send the image to print
Vector	Print or save the image in vector form
Save	Save the image in BMP or PNG format

> **Tip.** For a report it is more convenient to save the image in vector format: it can be scaled without quality loss and labels can be edited in a graphics editor.

Record video dialog

The dialog records a video of 3D model rotation for presentations and reports. It is opened with the **Options / Import/Export / Create video** command.



Record video window

Button	Purpose
Left button (**Start/Stop recording**)	Start recording; pressing again stops recording, saves the file and closes the dialog
Start/Stop rotation	Start or stop 3D model rotation

Immediately after the dialog opens, the 3D model starts rotating. The image of the 3D view window is recorded; the video is saved to the **Movie.avi** file in the program folder. The Xvid video codec must be installed in the system for recording.

> **Attention.** Each new recording overwrites the Movie.avi file, so rename or move a finished video immediately. Do not cover the 3D view window with other windows during recording.

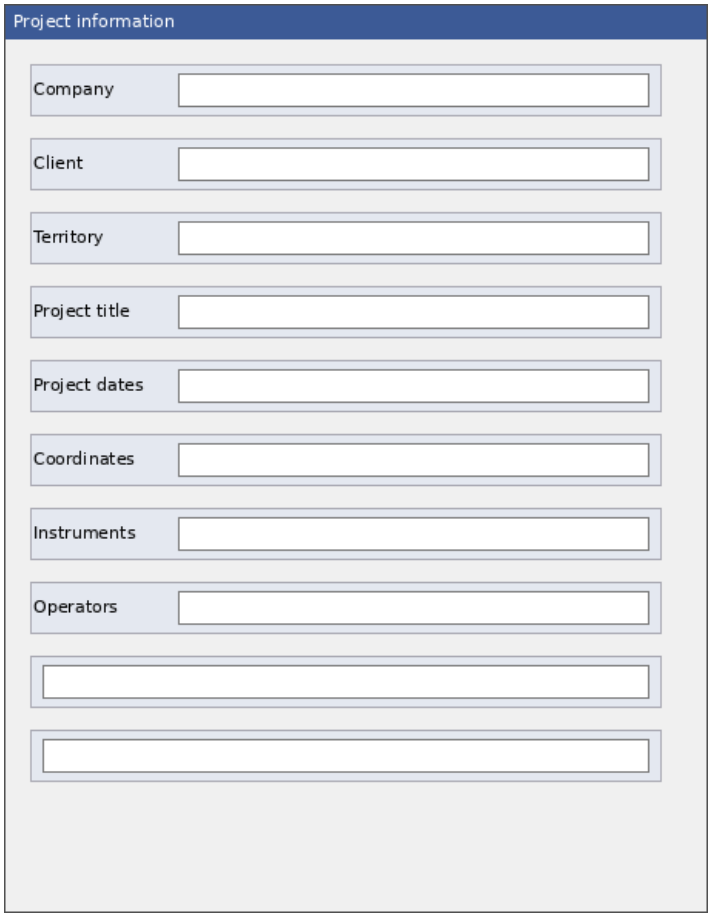
Workflow: recording a video

1. Set up the 3D image of the model: palette (via a model slice), cuts, view angle and scale.
2. Select **Options / Import/Export / Create video**: the model starts rotating.
3. If necessary, stop the rotation with the **Start/Stop rotation** button, choose the starting view and turn the rotation on again.
4. Press **Start/Stop recording**: recording starts.
5. After one or two full turns press **Start/Stop recording** again; the dialog closes.
6. Rename the Movie.avi file in the program folder.

Auxiliary windows

Project information window

The **Project information** window (**Options / Project information** command) stores information about the work: client, site, dates, instruments, personnel. The information is saved in the project file and is needed for documentation: opening the project a year later, one can immediately understand where the data came from and who acquired them.



Project information window

Field	Purpose
Company	Contractor organization
Client	Client
Territory	Territory, survey site
Project title	Project name
Project dates	Survey dates
Coordinates	Coordinates or coordinate system of the site
Instruments	Equipment used
Operators	Operators, personnel
Two bottom fields	Free fields for arbitrary information; a right click opens the auto-fill menu

Free field menu items:

Item	What is inserted
Geography info	Name of the locality found from the coordinates of the receiver centre (the project UTM zone is used)
Survey info	Survey summary, for example: “TEM survey, 12 sources (12 loops), 240 receivers, 5 lines, 30 times, 7150/7200 records, dB/dt”: the number of sources (of them loops), receivers, lines, times, the number of used/all records and the data type (B, dB/dt)

When the window opens, the saved information is loaded into the fields; when it closes, the entered values are written to the project (saved to disk when the project is saved).

Help info window

The **Help info** window shows brief help for a main menu item. To open it, right-click the item of interest in the opened main menu. The window shows the hint text of this item; the window stays on top of other windows and is updated when the next menu item is right-clicked.

Element	Purpose
Help text field	Description of the selected main menu item

Send message window

The **Send message** window (**Help / Send message to us** command) sends a message to the developers directly from the program: a question, an error report or a suggestion.



Send message window

Element	Purpose
NAME	Sender name
COMPANY	Organization
EMAIL	E-mail address for the reply (initial value “@”)
Messages not in English will be ignored	Reminder: messages are accepted only in English

Element	Purpose
Message field	Message text
Send	Close the window and send the message

After the window is closed, the name, organization and address are remembered (in a settings file with the .in extension next to the program executable) and filled in at the next opening. The message is sent if the address contains the "@" and "." characters: the program file name, filled fields and non-empty message lines are transmitted. If the address fails the check, the message "No email!" is shown.

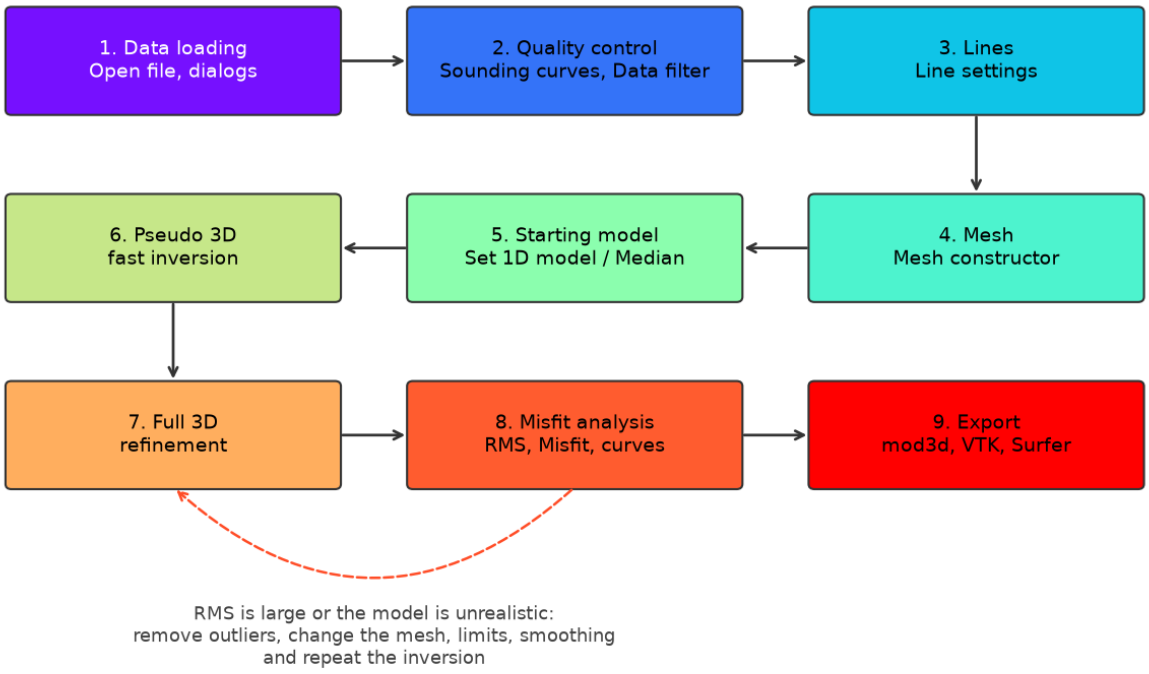
> **Tip.** When describing an error, specify the program version and the sequence of actions and attach (in a separate letter) the project or data file: the answer will be faster and more accurate.

Keys and mouse in graphic settings windows

Action	Combination	Where it works
Object context menu	Right mouse button outside the plot area	Pseudosection, map, data plots
Open Graphics setup	Shift + right mouse button on a curve	Graphics plot mode
Show only one curve	Left mouse button on a curve (hold)	Graphics plot mode
Axis menu (options / default / fix range)	Shift + right mouse button on an axis	All objects with axes
Fill interval / isoline colour	Right mouse button in a C_color / L_color cell	Edit levels
Fill / isoline palette	Right mouse button on the C_color / L_color header	Edit levels
Automatic dialog	Right mouse button on the Level header	Edit levels
Turn all isolines on / off	Left / right mouse button on the L_visible header	Edit levels
Add a slider	Ctrl + left mouse button in an empty place	Palette
Delete a slider	Ctrl + left mouse button on a slider; Delete for the selected one	Palette
Move a slider	Dragging with the left button	Palette
Moving an object on the sheet	Dragging with the left button	Print preview
Help on a menu item	Right mouse button on a main menu item	Main menu (Help info window)
Auto-fill of a free field	Right mouse button in the field	Project information

Typical work scenarios

This chapter collects short step-by-step scenarios. Each step refers to a window or menu item described in detail in the previous chapters. The scenarios need not be followed literally: they are a proven sequence of actions that helps not to miss important steps.



General scheme of processing and inversion in ZondTEM3D

Full cycle: from field data to a 3D model

1. **Open the data.** The button or **File / Open file**. In the list of file types select the format of your instrument. If there are several files (by lines or by loops), select them all at once: they will be combined into one data set.
2. **Answer the reader dialogs** (processing type, Curve selector, Amira, Geonics, Data units, UTM zone, depending on the format; see the “Opening data and projects” chapter).
3. **Check the coordinates.** If the file has no coordinates, the **Stations locations** window opens: load a coordinate file or paste coordinates from Excel. If the coordinates are in degrees, press **LL->XY**.
4. **Filter the data** in the **Data filter** window (if **File / Show data filter** is on): discard obviously uninformative early and late times, too weak signal, stations outside the site (User polygon).
5. **Set the lines** in the **Line settings** window. Lines define pseudosections and the list of lines on the quick access panel; for an areal survey automatic splitting is convenient.
6. **Build the mesh** in the **Mesh constructor** window. To start with, an automatic mesh is sufficient (the vertical mesh auto-selection button and an XY step close to the station spacing). Press **Apply**.
7. **Check data quality** (scenario below) and delete outliers.

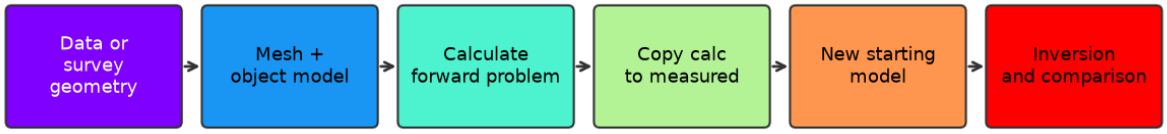
8. **Set the starting model.** By default the model is homogeneous. If a layered section is known (1D inversion in ZondTEM1D, a borehole), set it via **Options / Model / Set 1D model**; resistivity limits by depth are set there as well.
9. **Run the Pseudo 3D inversion.** Right-click the inversion button, select **Pseudo 3D inversion**, then left-click the button. The calculation takes seconds to minutes.
10. **Evaluate the result:** the misfit in the status bar, pseudosections of observed and calculated data, the misfit plan (**Options / Data / Data misfit**), curves of individual stations in the **Sounding curves** window.
11. **Run the Full 3D inversion.** Select **Full 3D inversion** in the same pop-up menu and start it. The inversion starts from the current model, i.e. from the Pseudo 3D result. If necessary, decrease the number of iterations in **Program setup** for the first trial run.
12. **Repeat the misfit analysis.** If individual stations are systematically not fitted, check their geometry (the **Station parameters** window), delete outliers with the command **Options / Extra / Remove data with big misfit** and continue the inversion.
13. **Save the project** (**File / Save file**, Project file type, *.t3d) and, if necessary, the project with calculated data.
14. **Export the result: Export mod3d**, **Export to Paraview**, **Draw to surfer**, images of sections and the 3D view (“Visualization and saving of results” chapter).

> **Tip.** Save intermediate models to the buffer (**Buffer / Model 1…4**) before each significant change of parameters. This allows quickly comparing inversion variants and returning to the best one.

Data quality control

1. Open the **Sounding curves** window with the button.
2. Browse the stations with the arrows or window buttons. Pay attention to sign changes, “steps”, sharp outliers at individual times, intervals where the signal drops to the noise level.
3. Delete obviously erroneous points: right click on a point, **Delete point**, or the **X** mode (deletion circle, the radius is changed with the mouse wheel). An erroneous deletion is undone with the button.
4. If a station is entirely uninformative, clear its **Invert** flag: it will remain on the plots but will not take part in the inversion.
5. Look at the plan and pseudosections at different times: a single station sharply different from its neighbours is a candidate for deletion or for geometry checking.
6. With strong noise at late times apply smoothing (**Options / Data / Smooth data**) or limit the time range.
7. Set the data norm (**Options / Inversion / Resolution / Data norm**): a threshold below which the signal is considered noise reduces the influence of late times with small amplitude.

Modelling and resolution check (synthetic test)



Synthetic test

A synthetic test answers the questions: “will a survey with such geometry see an object at such depth?” and “how reliably does the inversion recover a known model?”.

1. Open a real data file (or a file with the required geometry of loops and receivers) and build the mesh.
2. Set a layered background model (**Set 1D model**) and insert a local object (conductor or insulator) of the required size in the model editor.
3. Calculate the forward problem with the  button.
4. If there are no observed data, run **Options / Extra / Copy calc to measured**: the calculated response becomes “observed”. The item is available only when the project has no real observations; if the file contains field data, save the calculated data (**Save calculated data**, TDF format) and open this file as data.
5. If desired, add noise by editing the saved TDF file in an external program.
6. Save the object model to the buffer (**Buffer / Model 1**) to compare it with the result later.
7. Set a homogeneous or layered starting model without the object and run the inversion (Pseudo 3D, then Full 3D).
8. Compare the result with the original model (**Buffer / Open** window): the position, depth, contrast and size of the object.

Single-line survey (2D mode)

1. Open the data; leave one line in the **Line settings** window.
2. Turn on **Options / Model / 2D mode** (the mode is turned on manually).
3. In the **Mesh constructor** window select the **One line** survey type and build the mesh. With 2D mode on, cells across the line are combined into one inversion parameter, so the model remains homogeneous across the line; an edit in the editor is applied to all parallel sections.
4. Run the inversion as in the full cycle. Remember that objects to the side of the line will be projected into the section: 2D mode is applicable if the structures are elongated across the line.

Areal survey with several lines

1. Load all survey files at once or add them one by one with the **File / Append data** command.
2. Create lines in the **Line settings** window: automatically (by direction and step) or manually, specifying line points with the mouse. Stations not falling on any line are shown only on the plan.

3. If the receiver is offset relative to the loop centre and the offsets in the file are given in the local line system, run **Options / Extra / Align RX-TX along lines**.
4. Build a mesh with an XY step approximately equal to the station spacing; outside the site the mesh is coarsened automatically.
5. Start with Pseudo 3D: it quickly gives a general picture over the whole area and a starting model for Full 3D.
6. For Full 3D with a large number of loops, first check the calculation time of one forward problem; if necessary, thin the data (Data filter) or increase the mesh step outside the site.
7. Analyse the result on the plan (horizontal slices **Plane / XY**), on sections along lines and in the volume window (isosurfaces, Multi-slide).

Using a priori information

1. Load a background (geological section, seismics) onto a model slice: **Background 2D / Load background**, and georeference it.
2. Load borehole data (**Boreholes / Load borehole data**): they will be shown on sections and in 3D.
3. Set known resistivity values: via **Set 1D model** (layers with limits), by model build-in (**Build-in mod3d**, **Build-in mod2d's**) or manually in the model editor.
4. For reliably known cells set narrow Min/Max limits (**Cell setup** dialog or **Set 1D model**) so that the inversion does not lead them away from known values.
5. If the position of a boundary is known (bedrock top, groundwater level), build a mesh with a node plane at this depth and set the boundary in the **Options / Inversion / Set apriori boundaries** window. Boundaries are not written to the project: set them after opening the project before each inversion.
6. For a known layered section turn on **Options / Inversion / Optimization / Layered term** (start with **Pushing factor** = 1).
7. Run **Full 3D inversion** (boundaries and the layered term are used only in it) and compare the result with the a priori data.

Appendix A. File formats

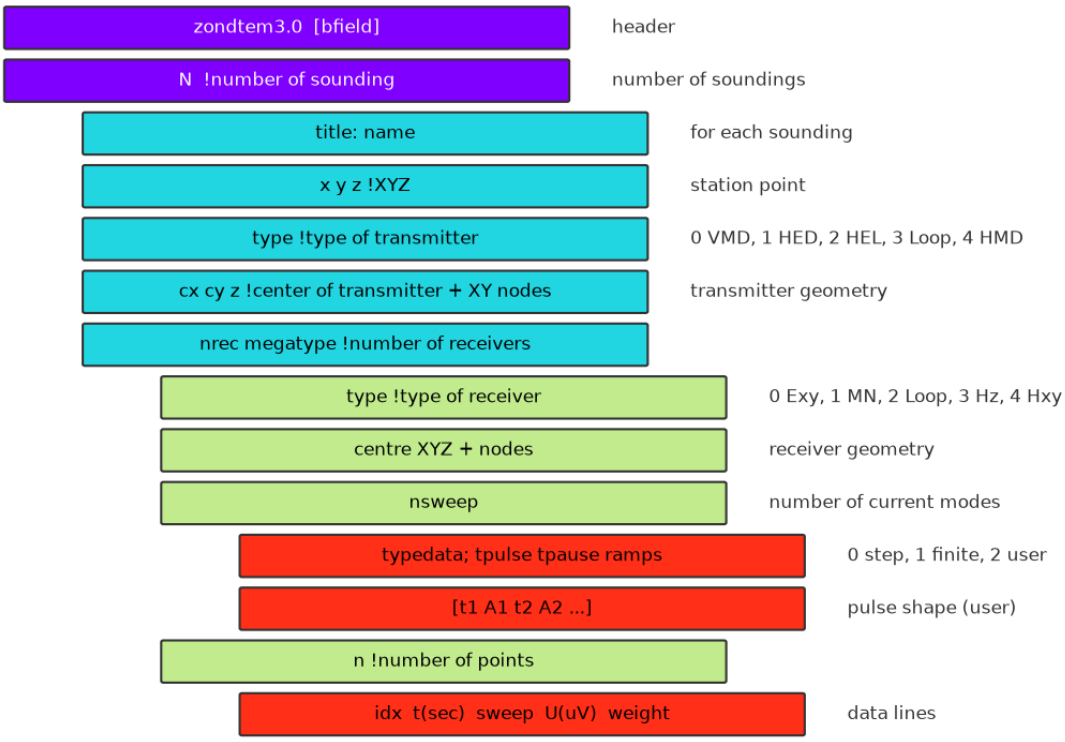
The appendix describes the text files that the program reads and writes. For each format, the command that creates or reads it is indicated, and a column table and a short example are given. General rules for all text files:

- separators are spaces or tabs (unless stated otherwise);
- the decimal separator is a point;
- coordinates and depths are in metres, time in seconds, resistivity in Ohm·m;
- lines containing fewer numbers than required are usually skipped (this allows keeping headers in the file).

TEM data

TDF format (zondtem3.0)

The own text data format of the ZondTEM programs. For each sounding it describes the transmitter (loop, line or dipole), one or several receivers, current modes and observed values. The file is created with the **Options / Import/Export / Save calculated data** command (calculated data) and read with the **File / Open file** command with the **ZondTEM1D field data** type (extensions `*.tdf`, `*.usf`, `*.avg`). The same format is used by ZondTEM1D. All lines allow a comment after the `!` sign: it is ignored when reading and serves as a hint for manual editing.



TDF file structure: block nesting

Line order:

Line	Contents	Explanation

Line	Contents	Explanation
1	`zondtem3.0` or `zondtem3.0 bfield`	Signature. The word `bfield` means that the file contains the B field, without it the derivative dB/dt
2	`N`	Number of soundings (stations)
For each sounding:		
3	`title: <name>`	Station name
4	`X Y Z`	Station point coordinates X, Y and its elevation
5	`type`	Transmitter type: 0, VMD (vertical magnetic dipole); 1, HED (horizontal electric dipole); 2, HEL (grounded line); 3, Loop; 4, HMD (horizontal magnetic dipole)
6	`cx cy z`	Transmitter centre X, Y and its height
7	`x1 y1 x2 y2 …`	For a loop and a line, plan coordinates of the vertices; for dipoles, centre X, Y and direction cosines
8	`nrec megatype`	Number of receivers and receiver group type (see below)
For each receiver:		
9	`type`	Receiver type: 0, Exy (electric dipole); 1, MN (receiving line); 2, Loop (receiver loop); 3, Hz (vertical magnetic dipole / induction coil); 4, Hxy (horizontal magnetic dipole). ZondTEM3D uses only types 2 and 3 (Z component); receivers of other types are skipped when reading
10	`cx cy z`	Receiver centre X, Y and its height
11	`x1 y1 …`	Vertices of the receiver loop/line or centre and direction cosines of the dipole
12	`nsweep`	Number of current modes (sweeps)
13	`typedata`	Pulse shape of the mode: 0, step; 1, finite pulse; 2, user-defined
14	`tpulse tpause ramp_front ramp_back`	Durations of the pulse, pause, ramp-on and ramp-off, s
15	`t1 A1 t2 A2 …`	Only for `typedata = 2`: pulse shape nodes (time, s; relative amplitude)
16	`n`	Number of records
17	`# t(sec) sweep U(uV) weight`	Header line (skipped)

Line	Contents	Explanation
18…	`i t sweep U weight`	Record number, time, s; current mode number; value; weight

The value **U** is written in μV/A (signal normalized by the transmitter current); for `bfield` files, the B field value normalized by the current. Weight 1 is an ordinary measurement; smaller values reduce the contribution of the record to the misfit.

The `megatype` parameter determines for which receivers current modes and records are written: with `megatype` = 0, for each receiver; with 1, 2, 5, 6, only for the first one (the others use the same times); with 3, for the first two. Receivers for which the block is not written are described only by lines 9–11.

Example: one station, 100×100 m loop, receiver is an induction coil at the centre, one current mode:

```
zondtem3.0
1 !number of sounding
title: ST001
500 1200 0 !XYZ coordinates
3 !type of transmitter/Loop
500 1200 0 !center of transmitter, XYZ
450 1150 550 1150 550 1250 450 1250  !XY nodes of loop
1 0 !number of receivers
3 !type of receiver/Hz dipole
500 1200 0 !center of receiver, XYZ
500 1200  !XY nodes
1 !number of sweep
1 !sweep1 Pulse 0 - step, 1 - finite, 2 - user
0.01 0.01 1E-5 5E-6 !tpulse, tpause, ramp front, ramp back (sec)
3 !number of points
#  t(sec) sweep  U(uV) weight
1 1E-5 1  1520.4  1
2 1E-4 1  48.71  1
3 1E-3 1  0.932  1
```

Data table (*.txt)

Created with the **File / Save file** command, type **Data table [*.txt]**. All receivers and times of the project are written, one line per record. The separator is a tab; a missing value is denoted by `*`. The file is convenient for building plots and maps in external programs (Excel, Surfer, Python).

№	Column	Units	Meaning
1	Rx	—	Receiver (station) name
2	X	m	X coordinate of the receiver centre
3	Y	m	Y coordinate of the receiver centre
4	H	m	Receiver height above the surface
5	Z	m	Absolute elevation of the receiver (topography + H)
6	Tx	—	Transmitter number (from 1)
7	Line	—	Line name or – if the receiver does not belong to a line

№	Column	Units	Meaning
8	Pos	m	Coordinate along the line
9	Time_s	s	Record time
10	dBdt	—	1, dB/dt data; 0, B field
11	Obs_uV/A	µV/A	Observed signal (* means a deleted point)
12	Calc_uV/A	µV/A	Calculated signal (* means no calculation)
13	RoaObs	Ohm·m	Apparent resistivity from observed data
14	RoaCalc	Ohm·m	Apparent resistivity from calculated data
15	Weight	—	Record weight

```
Rx     X       Y       H   Z       Tx  Line  Pos     Time_s   dBdt  Obs_uV/A  Calc_uV/A  Ro
aObs  RoaCalc  Weight
ST001  500     1200    0   152.3   1   L1    0       1E-05    1     1520.4    1498.2     85
.3    86.1     1
ST001  500     1200    0   152.3   1   L1    0       0.0001   1     48.71     50.02      92
.6    91.4     1
```

Station coordinate file

Read with the button of the **Stations locations** window. One station per line: station name and coordinates.

№	Column	Units	Meaning
1	Name	—	Station name (case-insensitive)
2	X	m (or degrees)	X coordinate (longitude)
3	Y	m (or degrees)	Y coordinate (latitude)
4	Z	m	Topography elevation (optional column)

A line is matched with a station first by the first word (name). If the first word does not match any name, a line is searched in which the station name occurs as a separate word. Stations absent in the file keep their previous coordinates. If the coordinates are given in degrees, press **LL->XY** after loading.

```
ST001  500.0  1200.0  152.3
ST002  550.0  1200.0  152.9
ST003  600.0  1200.0
```

Pulse shape file

Written and read with the buttons of the **Pulse shape** window. The first line is the number of nodes, then one line per node.

Line	Column 1	Column 2
1	N, the number of nodes	—
2 … N+1	Time, s (negative or 0; moment 0 is the end of the current turn-off)	Relative current amplitude (from −1.1 to 1.1; 1 is the steady-state current)

In the **Pulse shape** window table time is shown in microseconds, while the file stores it in seconds. At least three nodes are required.

```
4
-0.0100100 0
-0.0100000 1
-0.0000100 1
0 0
```

Project (*.t3d)

Binary project file: data with the whole geometry and current modes, deleted points and weights, mesh, model with limits, inversion and display parameters, lines, backgrounds, boreholes, model buffer. The **Project with calculated** type additionally saves calculated data, so after opening there is no need to solve the forward problem again. The project file is read only by ZondTEM3D; for exchange use TDF, MOD3D and data tables.

Mesh and topography

Topography file

Read with the **Import topography** command and written with the **Save topography** command of the menu opened by a right click on the topography button  of the **Mesh constructor** window. One point per line; the first three numbers of the line are used. Lines with fewer than three numbers (for example, a header) are skipped, as are points with repeated X, Y.

№	Column	Units	Meaning
1	X	m	X coordinate of the point
2	Y	m	Y coordinate of the point
3	Z	m	Absolute topography elevation (positive upwards)

```
1000.0  2000.0  152.3
1010.0  2000.0  152.9
1020.0  2000.0  153.4
```

When saving, the same three columns are written. If external topography is loaded, its points are saved, otherwise data points with known elevation.

Models: mod3d, mod2d, XYZV, XYZ slices

MOD3D (Export mod3d / Build-in mod3d)

Text file with the model mesh, parameter values and cell limits. Used for transferring the model between projects and ZOND programs.

- Created with the **Options / Import/Export / Export mod3d** command.
- Read with the **Options / Import/Export / Build-in mod3d** command. The **MOD3D model files** filter transfers the values of the current parameter into the existing mesh: cell to cell if the mesh sizes coincide, otherwise by interpolation. The **MOD3D mesh&mod files** filter replaces the project mesh with the mesh from the file (a standard extension is built around it) and transfers the parameter values.

The file consists of sections. The keyword stands at the beginning of the line, the data in the same line after a colon or in the following lines.

Section	Data	Meaning

Section	Data	Meaning
`software: <name>`	in the line	Program that created the file (for reference)
`units: m`	in the line	Coordinate units. When reading, any value other than `m` is treated as kilometres: coordinates are multiplied by 1000
`topoc: <c>`	in the line	Mesh topography coefficient c (see the formula below); usually from 0 to 1
`xyt_grid: Nx Ny Nz`	in the line; then Nx lines	Number of mesh nodes in X, Y and vertically. Line i contains Ny triples `X Y Z` of the top nodes (surface) for j = 1…Ny
`z_grid:`	next line	Nz depths of node planes from the surface, m; the first one equals 0
`paramnumb: n`	in the line	Number of parameters; ZondTEM3D writes 1 (resistivity)
`paramname i : <name>`	in the line	Name of parameter i: `rho`, resistivity (files of other ZOND programs may also contain `eta`, chargeability)
`islog i : yes/no`	in the line	Logarithmic scale of the parameter
`paramvalues i :`	(Nx−1)·(Ny−1) lines of Nz−1 numbers	Parameter values in cells
`parammin i :`	the same layout	Lower cell limits
`parammax i :`	the same layout	Upper cell limits; the pair 0 and 0 means no limits

Value layout. There is one cell fewer than nodes along each axis. The lines of the `paramvalues`, `parammin`, `parammax` blocks go by cell number in X (outer loop) and by cell number in Y (inner loop). Each line contains a vertical column of cells from top to bottom. Resistivity is given in Ohm·m.

When building in (**Build-in mod3d**), only parameter values (`paramvalues`) are read; the limits `parammin` and `parammax` are not read. The `rho` parameter is transferred; if it is absent in the file, the model does not change.

The position of node k under a surface point (i, j) is restored from the node plane depth z_k and the topography coefficient c:

$$Z_{ijk} = \min\left(Z_{ij}^{top} + z_k\left(1 + \frac{Z_{\max}^{top} - Z_{ij}^{top}}{z_N}\,c\right),\ Z_{\max}^{top} + z_k\right),$$

where Z_{ij}^{top} is the vertical surface coordinate from `xyt_grid` (in the program model system), $Z_{\max}^{top}$ is its largest value in the file, z_k are the `z_grid` values, z_N is the last of them. With $c = 0$ the mesh layers follow the topography, with $c = 1$ the mesh bottom is horizontal.

Example: a mesh of 3×3 nodes in plan and 4 nodes vertically, i.e. 2×2×3 cells. Numbers are shortened.

```
software: ZondTEM3D.
units: m
topoc:  5.0E-0001
xyt_grid: 3 3 4
 0.0  0.0  0.0  0.0  10.0  0.0  0.0  20.0  0.0
 10.0  0.0  0.0  10.0  10.0  -0.5  10.0  20.0  -1.0
 20.0  0.0  0.0  20.0  10.0  -1.0  20.0  20.0  -2.0
z_grid:
0.000 2.000 5.000 10.000
paramnumb: 1
paramname 1 : rho
islog 1 : yes
paramvalues 1 :
 100.0  100.0  300.0
 100.0  20.0  300.0
 100.0  100.0  300.0
 100.0  100.0  300.0
parammin 1 :
 0  0  100.0
 0  0  100.0
 0  0  100.0
 0  0  100.0
parammax 1 :
 0  0  1000.0
 0  0  1000.0
 0  0  1000.0
 0  0  1000.0
```

The second line of the `paramvalues` block refers to cells with number 1 in X and 2 in Y. Its second cell in depth has a resistivity of 20 Ohm·m (a conductive object). For the bottom cell layer a range of 100–1000 Ohm·m is written; the other cells have no limits.

MOD2D (Export mod2d / Build-in mod2d's)

Text file of a 2D model along a section line. Compatible with ZOND 2D interpretation programs (for example, ZondRes2D).

- Created with the **Options / Import/Export / Export mod2d** command for the current vertical slice (XZ or YZ). For a horizontal slice the file is not created.
- Read with the **Options / Import/Export / Build-in mod2d's** command: one or several files can be selected. Values of all files are interpolated into the whole 3D mesh of the current model. For each file the **Set rectangle** window opens with the coordinates of the line ends (**Left**, **Top**, **Right**, **Bottom**; by default from the `geocoord` line) and the **Insert topography** and **Use all points** check boxes.
- Viewed with the **Options / Import/Export / Plot MOD2D model** command in a separate window, without changing the project model.

Section	Data	Meaning
`software: <name>`	in the line	Program that created the file
`units: m`	in the line	Coordinate units; a value other than `m` is treated as kilometres when reading
`geocoord: x1 y1 x2 y2`	in the line	Plan coordinates of the start and end of the section line, m. The `gcoord` key is also read. If the line is absent, the line is assumed to lie along X

Section	Data	Meaning
		from the first to the last node of `x_grid` at Y = 0
`topoc: <c>`	in the line	Mesh topography coefficient, as in MOD3D
`x_grid:`	next line	Nx node coordinates along the line, m
`z_grid:`	next line	Nz node depths from the surface, m; the first one equals 0
`topo_grid:`	next line	Nx topography values at the `x_grid` nodes. When writing, relative to the minimum topography value of the slice
`topo_shift: <h>`	in the line	Topography shift. When writing, the minimum topography value of the slice
`paramnumb: n`	in the line	Number of parameters
`paramname i : <name>`	in the line	Parameter name (`rho`)
`islog i : yes/no`	in the line	Logarithmic scale
`minmaxCS i : <min> <max>`	in the line	Colour scale limits of the slice
`paramvalues i :`	Nx−1 lines of Nz−1 numbers	Values in cells: a line is a cell along the line, within a line the values by depth from top to bottom
`parammin i :`, `parammax i :`	the same layout	Lower and upper cell limits

Build-in features (**Build-in mod2d's**):

- the `rho` parameter is used; cell limits are not read;
- the plan position of each node is calculated along the `geocoord` line in proportion to its coordinate in `x_grid`;
- the value at a node is the average over the four adjacent section cells (for a logarithmic parameter, the average of logarithms);
- with the **Insert topography** check box the line topography is taken from the topography of the current 3D model. Without it, `topo_grid` and `topo_shift` from the file are used;
- without the **Use all points** check box, nodes falling outside the 3D model in plan are skipped.

```
software: ZondTEM3D.
units: m
geocoord: 0 50 200 50
topoc: 0
x_grid:
0 50 100 150 200
z_grid:
0 3 7 12
topo_grid:
2.1 1.5 0.8 0 0.4
topo_shift: 0
paramnumb: 1
paramname 1 : rho
islog 1 : yes
minmaxCS 1 : 10 1000
```

```
paramvalues 1 :
85.2 120.0 300.5
90.1 115.3 280.4
88.0 110.2 290.0
86.5 118.7 305.1
parammin 1 :
0 0 0
0 0 0
0 0 0
0 0 0
parammax 1 :
0 0 0
0 0 0
0 0 0
0 0 0
```

XYZV model

Model values at points in space. Used in two directions.

Writing: **File / Save file**, type **XYZV [*.dat]**. Values are written at model mesh nodes; the value at a node is the average over the adjacent cells (up to eight). The first line is a header starting with `#`; columns are separated by tabs, numbers are written in exponential form.

№	Column	Units	Meaning
1	X	m	X coordinate of the node
2	Y	m	Y coordinate of the node
3	Z	m	Vertical coordinate of the node in the program model system
4	rho	Ohm·m	Resistivity (in display units)
5	log_rho	—	Decimal logarithm of rho (only for a logarithmic parameter scale)

```
#X  Y   Z   rho log_rho
 0.0E+0000   0.0E+0000   1.0E+0002   1.523E+0002     2.18269E+0000
 5.0E+0000   0.0E+0000   1.0E+0002   1.489E+0002     2.17290E+0000
```

Reading: **Options / Import/Export / Build-in mod3d**, filter **XYZV model files**. Each line is a model point.

№	Column	Meaning
1	X	X coordinate, m
2	Y	Y coordinate, m
3	Z	Vertical coordinate in the program model system
4	V	Resistivity, Ohm·m
5…	—	Not used

Lines with fewer than four numbers (headers, comments) are skipped. More than five points are required. The value of each cell of the current mesh is calculated from the 12 nearest points of the file by the inverse distance method; for a parameter on a logarithmic scale logarithms are averaged.

```
X Y Z RHO
0.0   0.0   1.0   120.0
10.0  0.0   1.0   115.0
0.0   10.0  1.0   130.0
10.0  10.0  6.0   25.0
```

XYZ model slice

Created with the **File / Save file** command, type **XYZ [*.dat]**. The current slice shown in the model window is written at the slice mesh nodes. The value at a node is the average over the four adjacent slice cells. The separator is a tab.

Slice orientation	Column 1	Column 2	Then
Vertical along X (XZ)	X	Z	Parameter values
Vertical along Y (YZ)	Y	Z	Parameter values
Horizontal (XY)	X	Y	Parameter values

The header starts with `#`: `#X Z`, `#Y Z` or `#X Y`, then parameter names as in XYZV (`rho`, `log_rho`). Z is written in the program model system.

```
#X  Z   rho log_rho
 0.0E+0000   1.0E+0002   1.523E+0002     2.18269E+0000
 5.0E+0000   1.0E+0002   1.489E+0002     2.17290E+0000
```

Boreholes

Borehole data (.crt)

File with all borehole data of the project: lithological columns and logging curves with display parameters. Created and edited in the **Options / Boreholes / Create/Edit borehole data** window, loaded in this window or with the **Options / Boreholes / Load borehole data** command.

Logging data (.txt, .dat)

Loaded in the borehole editor. Each line contains depth and value.

№	Column	Meaning
1	Depth	Point depth (units are set in the **Settings** window of the editor)
2	Value	Value of the logging parameter

Lines with fewer than two numbers are skipped; at least four data lines are required.

```
Depth  Rho
0.5    120.4
1.0    118.2
1.5    95.7
2.0    60.1
```

Logging data (.las)

Standard LAS format. Depth and curve columns are selected in the column selection window; empty values (NULL) are excluded.

1D models (.mod1d)

1D models of ZOND programs (for example, 1D inversion results from ZondTEM1D or VES interpretation). Loaded with the **Options / Boreholes / Load borehole data** command; each model is shown as a pseudo-borehole.

Keyword	Meaning
`units`	If a line before the first model contains `km`, coordinates are treated as kilometres

Keyword	Meaning
`sounding_number`	Start of model descriptions
`title`	Start of the block of the next model (the text after the key is not used)
`…_xyz X Y [Z]`	Plan coordinates of the model point; the third number is not used
`par_rho` (`par_ro`)	Layer resistivities, Ohm·m
`par_eta` (`par_etha`)	Layer chargeabilities
`par_vs`, `par_vp`	Shear and compressional wave velocities
`par_grav`, `par_mag`	Layer densities and magnetic properties
`par_thick`	Layer thicknesses, m (one number fewer than parameter values)

Lines containing the word `range` are skipped. If the file has several parameters, the program offers to choose which one to show. The bottom layer is drawn to a depth 10 % greater than the depth of the last boundary.

```
units: m
sounding_number: 2
title: VES12
VES12_xyz 1250 3400 145
par_rho 120 35 800
par_thick 2.5 10
title: VES13
VES13_xyz 1300 3400 147
par_rho 110 40 750
par_thick 3 12
```

Lithology palette (.txt)

Text file with 11 “hatching colour” (brush color) records, one per palette cell of the borehole editor. Written and read in the **Create/Edit borehole data** window.

Palettes and .clr levels

LVL levels file (.lvl)

Levels file in the LVL2 format of Golden Software Surfer. Written with the **Save levels to file** button and read with the **Load levels from file** button of the **Edit levels** window. The first line is the `LVL2` signature, the second a field header starting with an apostrophe. Then one line per level; fields are separated by spaces, a quoted value with spaces is treated as one field.

№	Field	Writing	Reading
1	Level	Level value in display units	Level value
2	Flags	1 for the first level, 0 for the rest	Not used
3	LColor	Isoline colour `"Rr Gg Bb"`	Isoline colour; the field is searched by the `LColor` header, otherwise the third field
4	LStyle	`"Solid"`, the isoline is	Isoline visibility: hidden

№	Field	Writing	Reading
		visible; `"Invisible"`, hidden	with the `Invisible` value
5	LWidth	0	Not used
6	FFGColor	Fill colour `"Rr Gg Bb"`	Fill colour; the field is searched by the `FGColor` header, otherwise the sixth field
7–11	FBGColor, FPattern, FMode and others.	`"Black" "Solid" 1 1 1`	Not used

A colour can be written as `"R255 G0 B0"` or as a Surfer colour name (for example, `"Black"`). The fill colour of the interval above the last level is taken equal to the colour of the last level.

```
LVL2
'Level Flags LColor LStyle LWidth FFGColor FBGColor FPattern FMode
 10 1 "R0 G0 B0" "Solid" 0 "R0 G0 B255" "Black" "Solid" 1 1 1
 31.6 0 "R0 G0 B0" "Solid" 0 "R0 G255 B255" "Black" "Solid" 1 1 1
 100 0 "R0 G0 B0" "Invisible" 0 "R0 G255 B0" "Black" "Solid" 1 1 1
 316 0 "R0 G0 B0" "Solid" 0 "R255 G255 B0" "Black" "Solid" 1 1 1
 1000 0 "R0 G0 B0" "Solid" 0 "R255 G0 B0" "Black" "Solid" 1 1 1
```

Levels file .clr (Edit levels window)

Read with the **Load levels from file** button of the **Edit levels** window. Lines with at least four numbers are taken into account.

№	Column	Meaning
1	Level	Level value
2	R	Red component of the fill colour, 0–255
3	G	Green component, 0–255
4	B	Blue component, 0–255

The first number of the line is the level value itself, not a position in percent as in Surfer colour files. Isolines of all levels become black and visible.

```
10   0   0 255
100  0 255   0
1000 255 0   0
```

Palette file .csf

Written and read by the palette window (palette load and save buttons). The first line is the number of sliders N, then one line per slider.

Line	Column 1	Column 2
1	N, the number of sliders	—
2 … N+1	Colour code $R + 256\,G + 65536\,B$	Slider position on the scale, % (0–100)

Example of the “blue, light blue, green, yellow, red” palette:

```
5
16711680 0
16776960 25
65280 50
65535 75
255 100
```

Palette file .clr (palette window)

Read with the palette load button. Lines with at least four numbers are taken into account; the second, third and fourth numbers are the R, G, B components of the slider colour. The first number of the line is not used here: sliders are placed uniformly on the scale in line order. Surfer colour scale files, for example, have this form.

```
ColorMap 1 1
0    0   0 255
50   0 255   0
100 255  0   0
```

Export: Paraview, Oasis, Surfer/Voxler, XYZ, KML

VTK for Paraview (Export to Paraview)

Created with the **Options / Import/Export / Export to Paraview** command. The current 3D scene is written: slices, volume blocks, isosurfaces, cutting, transparency. Format: ASCII VTK, `DATASET UNSTRUCTURED_GRID`.

Block	Contents
`POINTS`	Vertex coordinates: X, Y and elevation Z
`CELLS`, `CELL_TYPES`	Scene polygons and segments
`CELL_DATA`	Scalar field `scene_group`, the number of the scene object group
`POINT_DATA`	The `display_rgba` field (`COLOR_SCALARS`, 4 components: colour and transparency as in ZondTEM3D) and the `model_value` field (parameter value in display units)

A `<name>_view.py` script for Paraview is written next to it. The script turns on colouring by `display_rgba` without mapping to a scale, so that the colours coincide with ZondTEM3D. To work with numerical values, select colouring by the `model_value` field in Paraview.

DXF for Oasis montaj (Export to Oasis)

Created with the **Options / Import/Export / Export to Oasis** command. DXF of AutoCAD 2004 version (`AC1018`), units are metres, coordinates are X, Y and absolute elevation.

Element	Contents
`3DFACE`	Surfaces; colour is the ACI index and the true RGB colour of the ZondTEM3D palette
`POLYLINE`	3D lines
`POINT`	Stations (`ELECTRODES` layer)
Layers	`MODEL` (slices), `MODEL_BLOCKS` (volume blocks), `GRID`, `ELECTRODES` (stations), `CUT`, `BACKGROUND`, `ISO_n`, `BOREHOLE_n`

Files for Surfer (Draw to surfer)

The **Options / Import/Export / Draw to surfer** command builds in Surfer maps of horizontal model slices. Intermediate data files and grids (`*.dat`, `*.grd`) and the script are written to the program folder and are intended only for this command.

voxout.dat and bla.dat (Draw to voxler)

The **Options / Import/Export / Draw to voxler** command writes to the program folder the value file `voxout.dat`, the mask file `bla.dat` and the script `voxout.bas`, and then runs the script via Voxler Scripter.

`voxout.dat` is a tab-separated text file: a header line, then one line per mesh node.

№	Column	Meaning
1, 2	X, Y	Node coordinates, m
3	Z	Vertical node coordinate (as in the XYZV file)
4	rho or log_rho	Parameter value at the node, the average over the eight adjacent cells. For a logarithmic parameter the decimal logarithm is written, and the column is named `log_<name>`

`bla.dat` is a cutting mask with the `#X Y Z B` header. It lists the points of the cut-off area (nodes and cell centres): X, Y, Z and flag 1. Used for clipping in Voxler.

```
#X  Y   Z   log_rho
 0.0E+0000   0.0E+0000   1.0E+0002   2.18E+0000
 5.0E+0000   0.0E+0000   1.0E+0002   2.17E+0000
```

Object XYZ file (Save XYZ file)

The **Save XYZ file** item of the pseudosection or map context menu saves the object points to a `*.dat` text file. The first line is the header `X Y Z` (or `X Y Z LOG10(Z)` for a logarithmic scale), then one line per point. The file is convenient for building maps in Surfer and other programs.

№	Column	Meaning
1	X	Horizontal coordinate of the point
2	Y	Vertical coordinate (pseudodepth) or the second map coordinate; with a reversed vertical axis it is written with a minus sign
3	Z	Parameter value in display units
4	LOG10(Z)	Decimal logarithm of the value (only for a logarithmic scale)

```
X Y Z LOG10(Z)
0.5 -0.75 120.4 2.0806
1.5 -0.75 98.1 1.9917
2.5 -1.50 143.2 2.1559
```

KML for Google Earth (Export to kml file)

The **Export to kml file** command of the map context menu writes two files with the same name: the background image (`*.png`) and a `*.kml` file. The item appears together with **Copy BG from model**; station coordinates must be given in UTM. The KML file contains an overlay (`GroundOverlay`) of the image on the terrain (`clampToGround`). The `LatLonBox` section contains the north, south, east and west boundaries of the image in degrees. They are recalculated from the UTM coordinates of the model corners in the zone selected for the satellite map.

```
<GroundOverlay>
  <name>Overlay on terrain</name>
  <Icon><href>map.png</href></Icon>
  <altitudeMode>clampToGround</altitudeMode>
  <LatLonBox>
    <north>55.75812345</north> <south>55.75012345</south>
    <east>37.62112345</east> <west>37.60812345</west>
  </LatLonBox>
</GroundOverlay>
```

Other service files

Background georeference file (.sec)

A two-line text file that links a raster image to coordinates. Read when loading a background:

- of a map in the line editor and on the plan (file type `*.sec` in the **Map** background open dialog);
- of vertical slices (**Options / Import/Export / Background 2D / Load background**).

Line	Contents
1	Path to the image file. If the file is not found at the given path, it is searched for in the folder of the `.sec` file
2	Four numbers, the coordinates of the upper left and lower right corners of the image: `x1 y1 x2 y2` for a map (X, Y in plan) or `X1 Z1 X2 Z2` for a vertical slice (distance and vertical coordinate)

For a vertical slice background, the word `km` in the second line means that the values are given in kilometres: they are multiplied by 1000.

Map background:

```
C:\Survey\site_map.bmp
5400100 6123500 5400600 6123000
```

Vertical slice background:

```
geol_profile1.png
0 0 450 -120
```

Appendix B. Keyboard shortcuts and mouse actions

The appendix collects all keyboard shortcuts and mouse actions of ZondTEM3D grouped by windows. Notation: LMB, RMB, MMB are the left, right and middle mouse buttons; “drag” means moving the mouse with a button pressed. Only the Alt+1, Alt+2, Alt+3 shortcuts are assigned to menu items; other commands are called via menus, panels and the mouse. In all value input dialogs Enter confirms the input, Esc cancels it (the previous value is kept).

Main window and graphic zones

Combination	Where it works	Action
RMB on a menu item	Main menu	Show the item hint in the Help info window (the window stays on top of the others)
Alt+1 / Alt+2 / Alt+3	Main window (Options / Model / Plane)	Model slice in the XZ / YZ / XY direction
LMB on the previous / next slice button	Quick access panel	Go to the neighbouring 2D slice in the current direction
LMB on the button Copy 2D section model	Quick access panel	Copy the current slice (all parameters and limits) to the buffer
LMB on the button Paste 2D model section	Quick access panel	Paste the slice from the buffer into the current slice or a group of slices
RMB on the button Paste 2D model section	Quick access panel	Paste mode menu: Current, Current&+, Current&-
LMB on the inversion button	Quick access panel	Start the selected inversion type; a repeated LMB during the calculation stops the inversion
RMB on the inversion button	Quick access panel	Select the inversion type: Pseudo 3D inversion / Full 3D inversion
RMB on the time list	Quick access panel	Set the number of neighbouring times (0–5) shown on the map together with the selected one
LMB on the button Sounding curves	Quick access panel	Show / hide the sounding curves window
Esc	Main window	Stop the Pseudo 3D inversion and other internal calculations (Full 3D is stopped with the inversion button)
LMB on a data point	Data map, pseudosections	Select a receiver: it is highlighted on the map and in 3D, the Sounding curves window shows its curves
Mouse wheel	Pseudosection in Graphics plot mode	Previous / next line
Dragging a splitter with LMB	Boundaries between main window zones	Resize the zones

Model and slice editor

Combination	Where it works	Action
Mouse wheel	Model slice (after clicking on it)	Next / previous slice
← / →	Main window, slice editor	Move the active cell left / right
↑ / ↓	Cursor over the model slice	Move the active cell up / down
↑ / ↓	Cursor over the slice colour scale	Increase / decrease the current value (brush) by a factor of 1.1
Insert	Slice editor	Assign the current value to the active cell (or to the selection, if the cell is selected)
Delete	Slice editor	Assign the host half-space value to the active cell (or the selection)
Z	Slice editor	Take the value of the active cell as the current one
X	Slice editor	Magic wand from the active cell: add a connected area of similar values to the selection
V	Slice editor	Clear selection
S	Slice editor	Select / deselect the active cell
LMB, click or drag	Slice, Editing mode	Assign the current value to cells; a click on a selected cell applies it to the whole selection
Shift+LMB / Shift+RMB	Slice, Editing mode	Increase / decrease the cell value by a given percentage (10 % by default); on a selected cell, the whole selection
LMB, drag	Slice, selection modes (free-form area, rectangle, ellipse)	New selection
Shift+LMB, drag	Slice, selection modes	Add an area to the existing selection
LMB	Slice, MagicWand mode	Select a connected area of cells with similar values
Ctrl+LMB, dragging a selected cell	Slice	Move the selection (the host half-space is left at the previous place)
Ctrl+RMB, dragging a selected cell	Slice	Copy the selection to a new place
Alt+RMB, drag	Slice	The context menu does not open, values do not change; in selection modes it continues the last selection from the previous starting point (contour points are added to a free-form area, a rectangle and an ellipse are rebuilt from the previous corner). After release the previous mode is restored
LMB on the colour scale	Slice	Select the current value and switch

Combination	Where it works	Action
		to Editing mode
RMB on a cell	Slice	Cell context menu (cell properties, editing and selection modes)
RMB on axes / on the colour scale	Slice	Image / colour scale context menu
LMB drag right-down / left-up	Slice, Zoom&Scroll mode	Zoom in an area / restore the scale
Ctrl+V	Start model and constraints, layer table	Paste a column of values from the clipboard
Dragging a line with LMB	Start model and constraints, plot	Change a layer value, boundary or depth
LMB / RMB	Set apriori boundaries, Add user defined polygon mode	Add a vertex / close the polygon
Mouse wheel	Set apriori boundaries, 3D view	Zoom

Sounding curves window

Combination	Where it works	Action
← / →	Sounding curves window	Previous / next receiver
Dragging a point with LMB	Curve plot	Change the point value
RMB on a point	Curve plot	Point menu: Delete point, Smooth point, Origin point, Delete all
Shift+RMB	Curve plot	Do not open the point menu
LMB / RMB on the smoothing button	Window panel	Smooth the current curve / all curves
LMB in X mode	Curve plot	Delete all points inside the circle
Mouse wheel in X mode	Curve plot	Deletion circle radius (4–200 pixels)
Dragging the red line with LMB	Data misfits (Remove data with big misfit)	Change the misfit threshold

Mesh constructor and Trapezoidal mesh

Combination	Where it works	Action
LMB on the plot	Mesh constructor, Remove XY row mode / Add XY row mode mode	Delete / add a row of nodes
RMB on the plot or on the mode button	Mesh constructor, row removal / addition modes	Switch the row direction (X / Y)
Press LMB on a node, release at a new place	Mesh constructor, Edit mesh nodes by mouse mode	Move the node
RMB on the node editing button	Mesh constructor	Correct mesh / Autocorrection menu
LMB / RMB on the Topography button	Mesh constructor	Show or hide the topography map / menu Import, Remove, Save, Satellite topography
RMB on the button Set trapezoidal	Mesh constructor	Reset the trapezoidal / simple mesh

Combination	Where it works	Action
mesh / Set simple mesh		
RMB on the Div number field	Mesh constructor (check box on)	Set the mesh step
Alt+LMB on the plot	Mesh constructor, Divide type = Manual, Axes ranges on, cursor in the field min or max	Take the boundary coordinate from the plot
LMB / wheel / RMB, drag	Mesh constructor, XYZ view	Rotate / zoom / pan
Shift+RMB on an axis	Mesh constructor, XYZ view	Axis setup
Ctrl+A	Trapezoidal mesh	New line (Add new line)
LMB / RMB, click or drag	Trapezoidal mesh, brush and strip	Assign stations to a line / remove the assignment
Mouse wheel	Trapezoidal mesh	Change the brush radius / strip width
Alt+wheel / Ctrl+wheel	Trapezoidal mesh, Simple grid maker	Stretch or compress the vertical / horizontal axis
LMB / RMB	Trapezoidal mesh, Add/remove node mode	Add / delete a point
Ctrl+LMB on the rotation button	Trapezoidal mesh, Simple grid maker	Rotation by 0.1° instead of 1°
RMB on the auto-search button	Trapezoidal mesh	Search direction menu
LMB / RMB on a line	Simple grid maker, Add remove node	Add / delete a line
LMB / RMB drag	Simple grid maker, Add fence mode	Insert N lines / delete lines in an interval
Mouse wheel while dragging	Simple grid maker, Add fence mode	Change the number of lines to insert

Lines, stations and data filter

Combination	Where it works	Action
LMB / RMB	Line settings, line adding mode	Add a vertex / last vertex, finish the line
LMB on a station	Line settings	Include or exclude the station from the active line
Ctrl+LMB on a station	Line settings	Highlight the receivers associated with the same transmitter loop
RMB on a station	Line settings	Change station X, Y, Z
Hold Alt	Line settings	Brush mode
Alt+LMB (click or drag) / Alt+RMB	Line settings, brush mode	Include / exclude stations with the brush
Mouse wheel	Line settings, brush mode	Brush radius
LMB / RMB on the Map button	Line settings	Background menu (Map / Topography) / hide the background
RMB on the line adding button	Line settings	Menu Set line coordinates, Cursor to near point, Invert line, Create lines

Combination	Where it works	Action
		auto
RMB on the zone map	UTM zone	Zone number (negative below the equator)
Ctrl+C / Ctrl+V	Stations locations	Copy cells / paste from the clipboard
RMB on an X or Y cell	Stations locations	Enter an increment and fill the column with a constant step
Ctrl+C / Ctrl+V	Pulse shape	Exchange of the pulse shape table with Excel
LMB / RMB	Data filter, User polygon tab	Add a vertex / last vertex, close the polygon

Pseudosections and maps

Combination	Where it works	Action
LMB drag right-down / left-up	Plan, pseudosections, plot plan	Zoom in an area / restore the original scale
RMB drag	Plan, pseudosections, plot plan	Pan the image
RMB outside the data field (on axes, margins)	Pseudosection, map, plot plan	Image context menu (Setup, Print preview, Save picture, etc.)
Shift+RMB on an axis	All plots with axes	Axis menu: Options, Default, Fix range
Shift+RMB on a point	Pseudosection	Point setup dialog (plot editor)
Mouse wheel	Point map	Point size (2–20)
LMB on a curve point (hold)	Graphics plot mode	Leave only this curve (observed and calculated) and mark the point; after release all curves are visible
Shift+RMB on a curve	Plot mode	Graphics setup window (group of observed or calculated curves)
Shift+LMB on a plot in the legend	Plot plan	Leave one plot (again: return the others)
Mouse wheel over the legend	Plot plan	Scroll the group of selected curves

The plan and pseudosection windows receive focus when the cursor hovers over them, so the mouse wheel acts in them immediately.

3D volume window

Combination	Where it works	Action
LMB drag	Volume model	Rotation
MMB drag	Volume model	Pan the image (measured from the press point: each new drag starts from the unshifted position)
Mouse wheel	Volume model	Scale 1–300 %, step 2
RMB	Volume model	Open the 3D settings window

Combination	Where it works	Action
Dragging a point with LMB	Opacity vs parameter	Change opacity at a point
Ctrl+drag a point	Opacity vs parameter	Set the same value for all points to the left / right
RMB	Opacity vs parameter	Set the level for all points
LMB / RMB	Set contour of cutting	Add a vertex / close the contour

Graphics and print windows

Combination	Where it works	Action
RMB on the upper / lower half of a C_color cell	Edit levels	Fill interval colour above / below the level
RMB on an L_color cell	Edit levels	Isoline colour
RMB on the C_color / L_color header	Edit levels	Fill / isoline palette
RMB on the Level header	Edit levels	Automatic dialog (uniform levels from minimum and maximum)
LMB / RMB on the L_visible header	Edit levels	Turn all isolines on / off
Select a Level cell and type a number	Edit levels	Set a level value
LMB on a slider / LMB drag	Palette	Select / move a slider
Ctrl+LMB in an empty place above the scale	Palette	Add a slider
Ctrl+LMB on a slider; Delete	Palette	Delete a slider (Delete: the selected one)
LMB drag of an object	Print preview	Move an object on the sheet
LMB drag of vertical lines	Model histogram	Set the colour scale limits

Other windows

Combination	Where it works	Action
RMB in a field	Project information, two bottom fields	Insert Geography info / Survey info
LMB drag of boundaries	Buffer / Open	Resize areas
RMB on a cell of the X or Y column	Program setup, Options, Cell grouping table	Set the group width for the layer and all underlying ones
Ctrl+V	Borehole editor, column H	Paste layer thicknesses
Repeated LMB / RMB on a cell of column C	Borehole editor	Hatching and colour editor / Exchange P&C, P, C menu
RMB on a layer	Borehole editor, graphic window	Layer hatching editor
LMB / RMB on a palette cell	Borehole editor	Apply / store a palette cell

Appendix C. Program files and modules

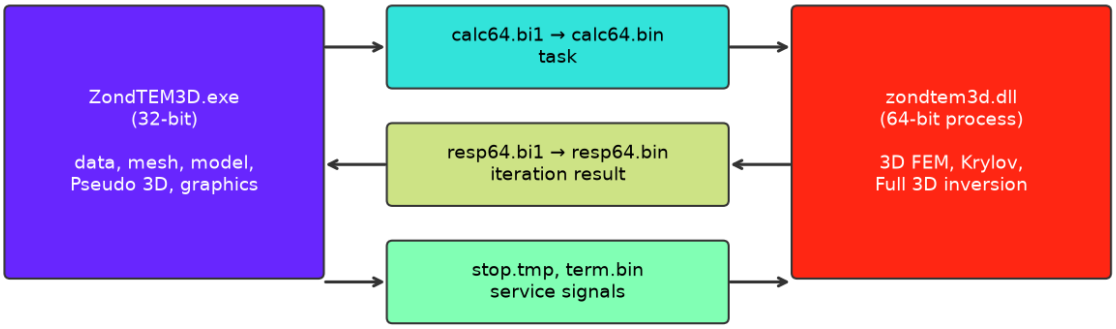
Files in the program folder

File	Purpose
`ZondTEM3D.exe`	Main program: interface, data, mesh, model, Pseudo 3D inversion, visualization
`zondtem3d.dll`	64-bit computational module: 3D forward problem by the finite element method (rational Krylov approximations) and Full 3D inversion. Despite the .dll extension, it is a separate 64-bit executable process that the main program launches at startup and keeps in the background (visible in the task manager). If the file is missing, the message “zondtem3d.dll not found” is shown at startup, and the forward problem and Full 3D are unavailable
`ZondTEM3D.in`	Program settings file (window sizes, recent files, display parameters). If it is deleted, the program starts with default settings
`ZondTEM3D_eng.pdf`	Manual opened with the **Help / Manual** command

Temporary exchange files

The main program and the computational module exchange data through temporary files in the program folder. Therefore the program folder must be writable. The files are created and deleted automatically; they must not be deleted manually during calculation.

File	Purpose
`calc64.bi1` → `calc64.bin`	Calculation task: mesh, model, station geometry, times, data and inversion parameters. It is first written under the name `calc64.bi1` and then renamed; this is the signal for the module to start the calculation
`resp64.bi1` → `resp64.bin`	Result: calculated data, model of the current iteration, misfit. Read by the main program after each iteration
`stop.tmp`	Stop signal for the module; created with the **Help / Restart library** command if the module process could not be terminated immediately. Stopping Full 3D with the inversion button terminates the module process and restarts it
`term.bin`	Service synchronization file at computational module startup
`voxout.dat`, `bla.dat`, `voxout.bas`, `box.dat`, `tmp.bas`, `*.grd`	Intermediate export files for Voxler and Surfer (Appendix A)



Data exchange between the main program and the computational module

If the computational module “hangs” (for example, after an abnormal termination), run **Help / Restart library**: the program will stop the module and delete the temporary files.

Program modules and windows

The table lists all program windows, their purpose and how they are called. A detailed description is in the indicated chapters.

Window	Purpose	How to open	Chapter
ZondTEM3D (main window)	Data map, pseudosections, 3D view, model editor	—	Main window
Curve selector, **Amira**, **Geonics survey**, **Data units**, **UTM zone**	Clarification of format parameters when reading a file: curve selection, units, geometry, UTM zone	Automatically when opening files of some instruments	Opening data
Stations locations	Input and loading of station coordinates	Automatically if there are no coordinates in the file	Opening data
Data filter	Data selection by time, signal, apparent resistivity and polygon	On opening (if **File / Show data filter** is on)	Opening data
Line settings and **Settings** (auto-build)	Survey lines	**Options / Set survey lines**	Opening data
Station parameters, **Pulse shape**	Transmitter and receiver geometry, current modes and waveform of a station	Button on the quick access panel	Opening data
Sounding curves	Receiver sounding curves, point editing, Invert flag	Button on the quick access panel	Viewing and processing data
Smoothing settings	Data smoothing in time and between stations	**Options / Data / Smooth data**	Viewing and processing data
Data misfits	Removing data with large misfit	**Options / Extra / Remove data with big misfit**	Viewing and processing data
Mesh constructor, **Trapezoidal mesh**, **Simple grid maker**	Building the model mesh	Automatically after data loading; **Options / Mesh constructor**	Mesh constructor
Start model and	Layered starting model	**Options / Model / Set 1D**	Model

Window	Purpose	How to open	Chapter
constraints	and limits by depth	**model**	
Cell setup	Value and limits of a cell or selection	Model editor menu	Model
Set apriori boundaries	Polygons of a priori boundaries	**Options / Inversion / Set apriori boundaries**	Model
Program setup	Inversion parameters, cell grouping	Button on the quick access panel	Forward problem and inversion
Extra layered term	Parameters of the "layered" term	**Options / Inversion / Optimization / Layered term**	Forward problem and inversion
RMS vs iteration	Inversion progress	**Options / Inversion / Show progress**	Forward problem and inversion
Set rectangle	Line position of the MOD2D model to build in	**Build-in mod2d's**	A priori information
Corners	Corner coordinates of the 3D background	3D view menu	A priori information
Add borehole data, **Locations**	Borehole data and their coordinates	**Options / Boreholes**	A priori information
3D settings, **Opacity vs parameter value**, **Smooth/raster model**, **Set contour of cutting**	3D view setup, transparency, smoothing and cutting of the model	Right button in the 3D view; **Options / Cutting** menu	Visualization and saving
Sections from buffer	Comparison of buffer models	**Buffer / Open**	Visualization and saving
Record video	Recording a 3D view video	**Options / Import/Export / Create video**	Visualization and saving
Model setup, **Contour-section setup**, **Graphics setup**, **Edit levels**, **Automatic**, **Color palette**, **Picture settings**	Setup of images, scales, levels and palettes	Image context menus	Graphic settings
Histogram	Histogram of model values	**Options / Model / Model histogram**	Graphic settings
Project information, **Help info**, **Send message**	Project information, menu help, message to developers	**Options** and **Help** menus; right button on a menu item	Auxiliary windows